Natural Sciences Tripos Part II

MATERIALS SCIENCE

FEM Practical Booklet

Name............................ College..........................

Dr J. Dean

Lent 2014-15

Athena SWAN Bronze Award

UNIVERSITY OF CAMBRIDGE
Department of Materials Science and Metallurgy

Contents

ABAQUS

Abaqus/CAE is a complete modelling environment that provides a simple, consistent interface for creating, submitting, monitoring, and evaluating results from **Abaqus/Standard** and **Abaqus/Explicit** simulations. Abaqus/CAE is divided into modules, where each module defines a logical aspect of the modeling process; for example, defining the geometry, defining material properties, and generating a mesh. As you move from module to module, you build the model from which Abaqus/CAE generates an input file that you submit to the Abaqus/Standard or Abaqus/Explicit analysis product. The analysis product performs the analysis, sends information to Abaqus/CAE to allow you to monitor the progress of the job, and generates an output database. Finally, you use the Visualization module of Abaqus/CAE to read the output database and view the results of your analysis.

This lecture includes a general introduction to the ABAQUS Graphical User Interface (GUI), an introduction to model formulation and the concept of Modules, discretisation and meshing techniques, boundary condition specification, job submission, results visualisation and analysis checks (convergence etc.). These features will be demonstrated using a finite element model of indentation – Fig.1.
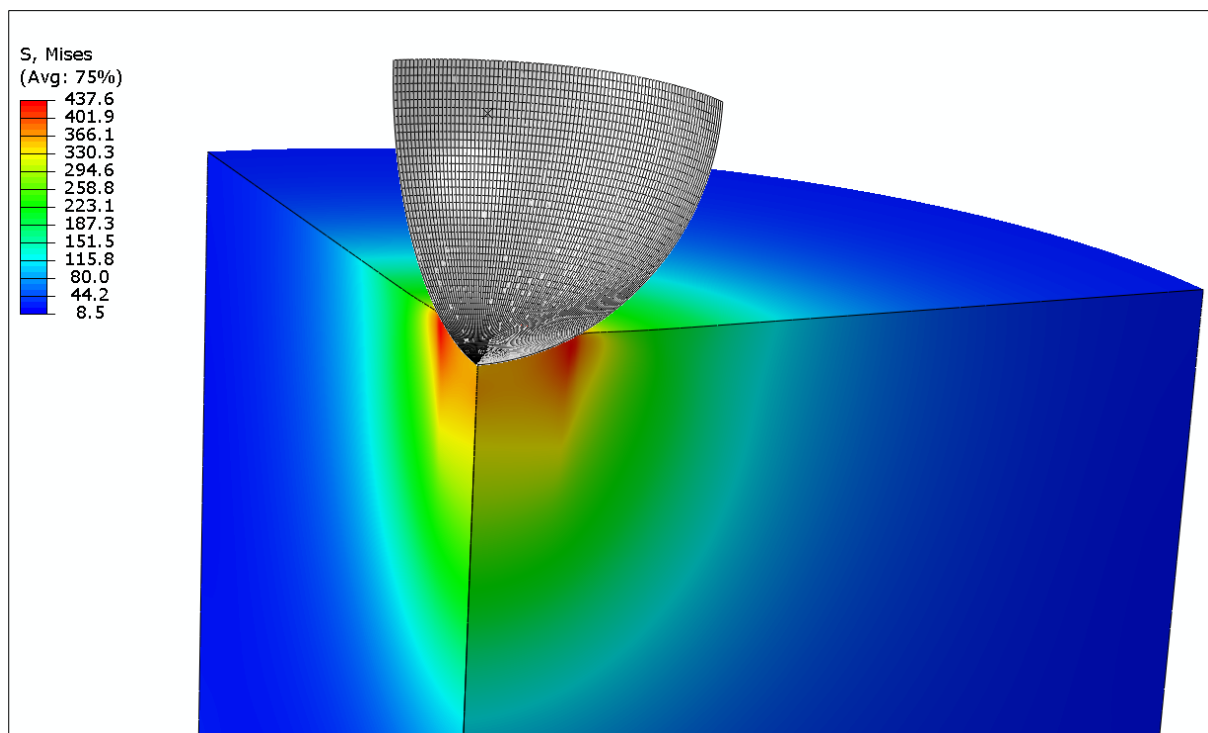
Figure 1: Predicted contours of von-Mises stress in a copper specimen indented with a rigid, spherical indenter (25 μm diameter)

Components of an Abaqus Model

Modules

Abaqus/CAE is divided into functional units called modules. Each module contains only the tools that are relevant to a specific portion of the modeling task. For example, the Mesh module contains only the tools needed to create the meshes, while the Job module contains only the tools used to create, edit, submit, and monitor analysis jobs. Abaqus/Viewer is a subset of Abaqus/CAE that contains only the Visualization module.

You can select a module from the **Module** list in the context bar. Alternatively, you can select a module by switching to the context of a selected object in the Model Tree. The order of the modules in the menu and in the Model Tree corresponds to the logical sequence you follow to create a model. In many circumstances you must follow this natural progression to complete a modelling task; for example, you must create parts before you create an assembly. Although the order of the modules follows a logical sequence, Abaqus/CAE allows you to select any module at any time, regardless of the state of your model.

The available modules in ABAQUS/CAE are the part, property, assembly, step, interaction, load, mesh, job, and visualisation modules.
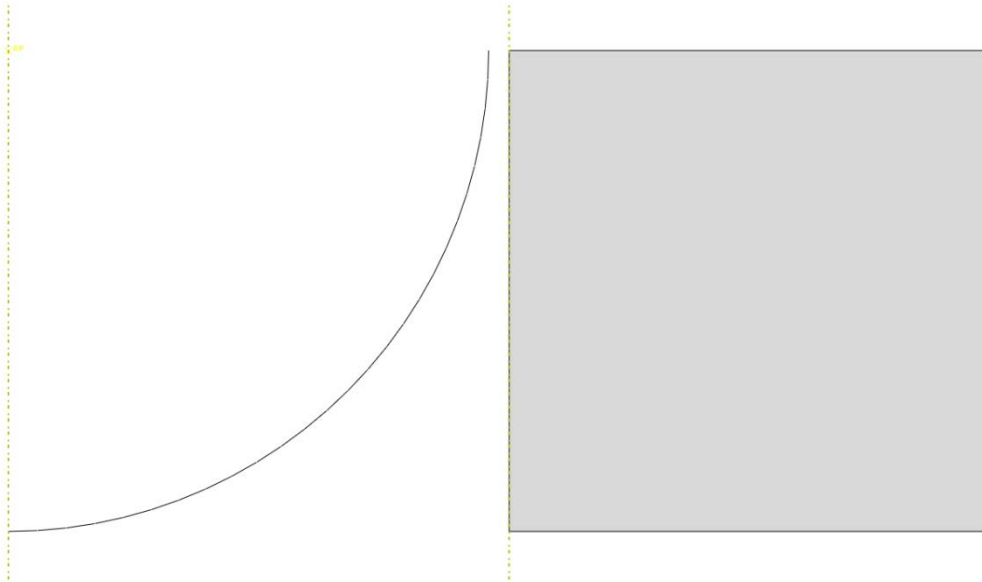
Parts

You use the Part module to create, edit, and manage the parts in the current model. Abaqus/CAE stores each part in the form of an ordered list of features. The parameters that define each feature—extruded depth, hole diameter, sweep path, etc. — combine to define the geometry of the part.

The Part module allows you to do the following:

- Create deformable, discrete rigid, analytical rigid or Eulerian parts. The part tools also allow you to edit and manipulate the existing parts defined in the current model.

- Create the features—solids, shells, wires, cuts, and rounds—that define the geometry of the part.

- Use the Feature Manipulation toolset to edit, delete, suppress, resume, and regenerate a part's features.

- Assign the reference point to a rigid part.

- Use the Sketcher to create, edit, and manage the two-dimensional sketches that form the profile of a part's features. These profiles can be extruded, revolved, or

swept to create part geometry; or they can be used directly to form a planar or axisymmetric part.

- Use the Set toolset, the Partition toolset, and the Datum toolset. These toolsets operate on the part in the current viewport and allow you to create sets, partitions, and datum geometry, respectively.

Figure 2: Component parts in the indentation model

Property

- Define materials.

- Define beam section profiles.

- Define sections.

- Assign sections, orientations, normals, and tangents to parts.

- Define composite layups.

- Define skin reinforcement.

- Define inertia (point mass, rotary inertia, and heat capacitance) on a part.

- Define springs and dashpots between two points or between a point and ground.

Material Models

You can define a number of different material behaviours in the Abaqus material editor, from simple elastic material behaviour to more complicated, highly non-linear constitutive behaviour. Material behaviors fall into the following general categories:

- general properties (material damping, density, thermal expansion);

- elastic mechanical properties;

- inelastic mechanical properties;

- thermal properties;

- acoustic properties;

- hydrostatic fluid properties;

- equations of state;

- mass diffusion properties;

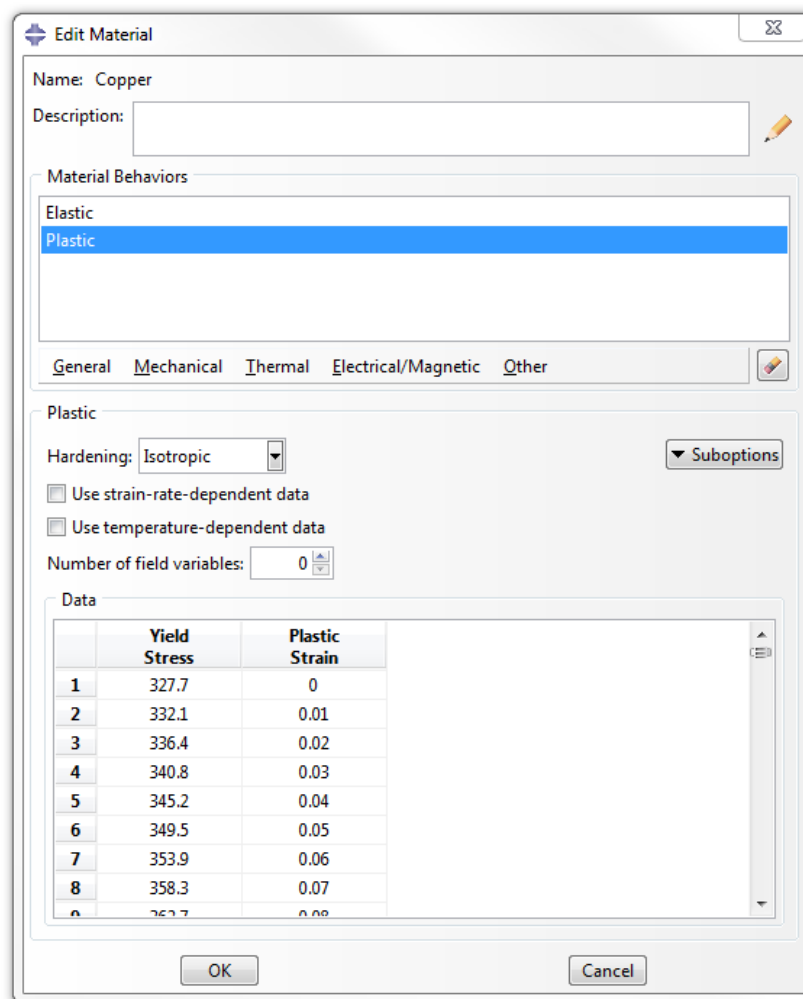- electrical properties;

- pore fluid flow properties.

Figure 3: Material editor in Abaqus

The material library in Abaqus is intended to provide comprehensive coverage of linear and nonlinear, isotropic and anisotropic material behaviours. The use of numerical integration in the elements, including numerical integration across the cross-sections of shells and beams, provides the flexibility to analyze the most complex composite structures.

Some of the mechanical behaviors offered are mutually exclusive: such behaviors cannot appear together in a single material definition. Some behaviors require the presence of other behaviors; for example, plasticity requires linear elasticity.

A material definition can include behaviors that are not meaningful for the elements or analysis in which the material is being used. Such behaviors will be ignored. For example, a material definition can include heat transfer properties (conductivity, specific heat) as well as stress-strain properties (elastic moduli, yield stress, etc). When this material definition is used with uncoupled stress/displacement elements, the heat transfer properties are ignored by Abaqus; when it is used with heat transfer elements, the mechanical strength properties are ignored. This capability allows you to develop complete material definitions and use them in any analysis.

For a comprehensive introduction on how to define material property data, students are advised to consult section 17.1.2 of the ABAQUS user manual, 'Material Data Definition'.

Units

Abaqus has no units built into it except for rotation and angle measures. Therefore, the units chosen must be self-consistent, which means that derived units of the chosen system can be expressed in terms of the fundamental units without conversion factors.

International System of units (SI)

The International System of units (SI) is an example of a self-consistent set of units. The fundamental units in the SI system are length in meters (m), mass in kilograms (kg), time in seconds (s), temperature in degrees Kelvin (K), and electric current in Amperes (A). The units of secondary or derived quantities are based on these fundamental units. An example of a derived unit is the unit of force. A unit of force in the SI system is called a Newton (N):

1 Newton = 1 kg m s^{-2}

Similarly, a unit of electrical charge in the SI system is called a Coulomb (C):

1 Coulomb = 1 A s

Another example is the unit of energy, called a Joule (J):

1 Joule = 1 N m = 1 A Volt s = 1 kg m^2 s^{-2}

The unit of electrical potential in the SI system is the Volt, which is chosen such that

1 Joule = 1 Volt C = 1 Volt A s

Sometimes the standard units are not convenient to work with. For example, Young's modulus is frequently specified in terms of Mega-Pascals (MPa) (or, equivalently, N/mm^2), where 1 Pascal = 1 N/m^2. In this case the fundamental units could be tonnes (1 tonne = 1000 kilograms), millimeters, and seconds.

*Three sets of consistent units can be found at the back of this handout.

<u>Rotation and angle measures</u>

In Abaqus rotational degrees of freedom are expressed in radians, and all other angle measures are expressed in degrees (for example, phase angles).

<u> Assembly</u>

When you create a part, it exists in its own coordinate system, independent of other parts in the model. In contrast, you use the Assembly module to create instances of your parts and to position the instances relative to each other in a global coordinate system, thus creating the assembly. You position part instances by sequentially applying position constraints that align selected faces, edges, or vertices or by applying simple translations and rotations. Move the parts about using the translate features.
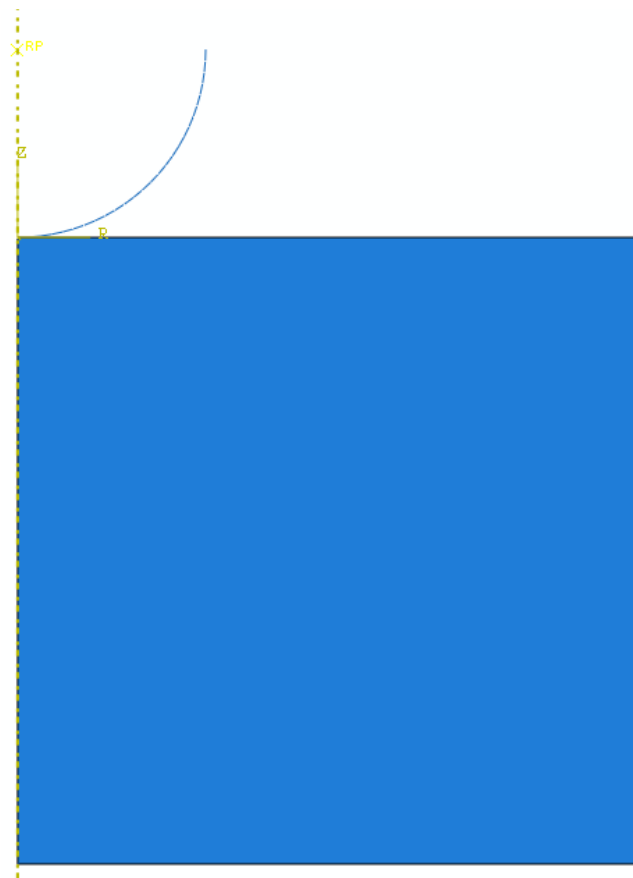
Figure 4: Assembly of part instances

An instance maintains its association with the original part. If the geometry of a part changes, Abaqus/CAE automatically updates all instances of the part to reflect these changes. You cannot edit the geometry of a part instance directly.

A model can contain many parts, and a part can be instanced many times in the assembly; however, a model contains only one assembly. Loads, boundary conditions, predefined fields, and meshes are all applied to the assembly. Even if your model consists of only a single part, you must still create an assembly that consists of just a single instance of that part.

A part instance can be thought of as a representation of the original part. You can create either independent or dependent part instances. An independent instance is effectively a copy of the part. A dependent instance is only a pointer to the part, partition, or virtual topology; and as a result, you cannot mesh a dependent instance. However, you can mesh the original part from which the instance was derived, in which case Abaqus/CAE applies the same mesh to each dependent instance of the part. (Note that dependent part instances are less expensive computationally - although not much!)

Step

You can use the Step module to perform the following tasks:

- Create analysis steps.

- Specify output requests.

- Specify adaptive meshing.

- Specify analysis controls.

Create analysis steps

- Within a model you define a sequence of one or more analysis steps. The step sequence provides a convenient way to capture changes in the loading and boundary conditions of the model, changes in the way parts of the model interact with each other, the removal or addition of parts, and any other changes that may occur in the model during the course of the analysis. In addition, steps allow you to change the analysis procedure, the data output, and various controls. You can also use steps to define linear perturbation analyses about nonlinear base states. You can use the replace function to change the analysis procedure of an existing step.

Specify output requests

- Abaqus writes output from the analysis to the output database; you specify the output by creating output requests that are propagated to subsequent analysis steps. An output request defines which variables will be output during an analysis

step, from which region of the model they will be output, and at what rate they will be output. See output requests for the model. For example, you might request output of the entire model's displacement field at the end of a step and also request the history of a reaction force at a restrained point.

Specify adaptive meshing

- You can define adaptive mesh regions and specify controls for adaptive meshing in those regions.

Specify analysis controls

- You can customize general solution controls and solver controls.

<u>Interaction</u>

You can use the Interaction module to define the following:

- Contact interactions.

- Elastic foundations.

- Cavity radiation.

- Thermal film conditions.

- Radiation to and from the ambient environment.

- Incident waves.

- Acoustic impedance.

- Cyclic symmetry.

- A user-defined actuator/sensor interaction.

- Tie constraints.

- Rigid body constraints.

- Display body constraints.

- Coupling constraints.

- Shell-to-solid coupling constraints.

- Embedded region constraints.

- Equation constraints.

- Connector section assignments.

- Inertia.

- Cracks.

- Springs and dashpots.

Interactions are *a step-dependent object,* which means that when you define them, you must indicate in which steps of the analysis they are active. For example, you can define film and radiation conditions on a surface only during a heat transfer, coupled temperature-displacement, or coupled thermal-electrical step. Similarly, you can define an interaction with a user-defined actuator/sensor only during the initial step.

The Set and Surface toolsets in the Interaction module allow you to define and name regions of your model to which you would like interactions and constraints applied. You can use the Amplitude toolset to define variations in some interaction attributes over the course of the analysis. The Analytical Field toolset allows you to create analytical fields that you can use to define spatially varying parameters for selected interactions. The Reference Point toolset allows you to define reference points that are used in constraints and creating assembly-level wire features.

Abaqus/CAE does not recognize mechanical contact between part instances or regions of an assembly unless that contact is specified in the Interaction module; the mere physical proximity of two surfaces in an assembly is not enough to indicate any type of interaction between the surfaces.

Load

There are many different load types that you can define in ABAQUS. Prescribed conditions in Abaqus/CAE are *a step-dependent object,* which means that you must specify the analysis steps in which they are active.

You can use the load, boundary condition, and predefined field managers to view and manipulate the stepwise history of prescribed conditions. You can also use the Step list located in the context bar to specify the steps in which new loads, boundary conditions, and predefined fields become active by default.

You can use the Amplitude toolset to specify complicated time or frequency dependencies that can be applied to prescribed conditions. The Set and Surface toolsets in the Load module allow you to define and name regions of your model to which you would like to apply prescribed conditions. The Analytical Field toolset and the Discrete Field toolset allow you to create fields that you can use to define spatially varying parameters for selected prescribed conditions.

Load cases are sets of loads and boundary conditions used to define a particular loading condition. You can create load cases in static perturbation and steady-state dynamic, direct steps.
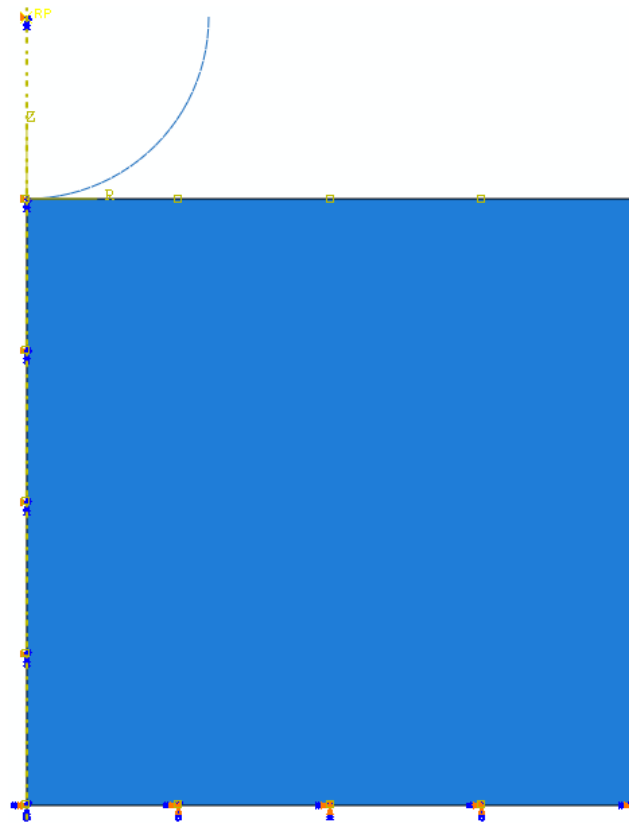
*Figure 5: Assembly of part instances with applied boundary conditions*

Mesh

The Mesh module allows you to generate meshes on parts and assemblies created within Abaqus/CAE. Various levels of automation and control are available so that you can create a mesh that meets the needs of your analysis. As with creating parts and assemblies, the process of assigning mesh attributes to the model—such as seeds, mesh techniques, and element types—is feature based. As a result you can modify the parameters that define a part or an assembly, and the mesh attributes that you specified within the Mesh module are regenerated automatically.

The Mesh module provides the following features:

- Tools for prescribing mesh density at local and global levels.

- Model coloring that indicates the meshing technique assigned to each region in the model.

- A variety of mesh controls, such as:

- Element shape

- Meshing technique

- Meshing algorithm

- Adaptive re-meshing rules

- A tool for assigning Abaqus/Standard and Abaqus/Explicit element types to mesh elements. The elements can belong either to a model that you created or to an orphan mesh.

- A tool for verifying mesh quality.

- Tools for refining the mesh and for improving the mesh quality.

- Tools for saving the meshed assembly or selected part instances as an orphan mesh part
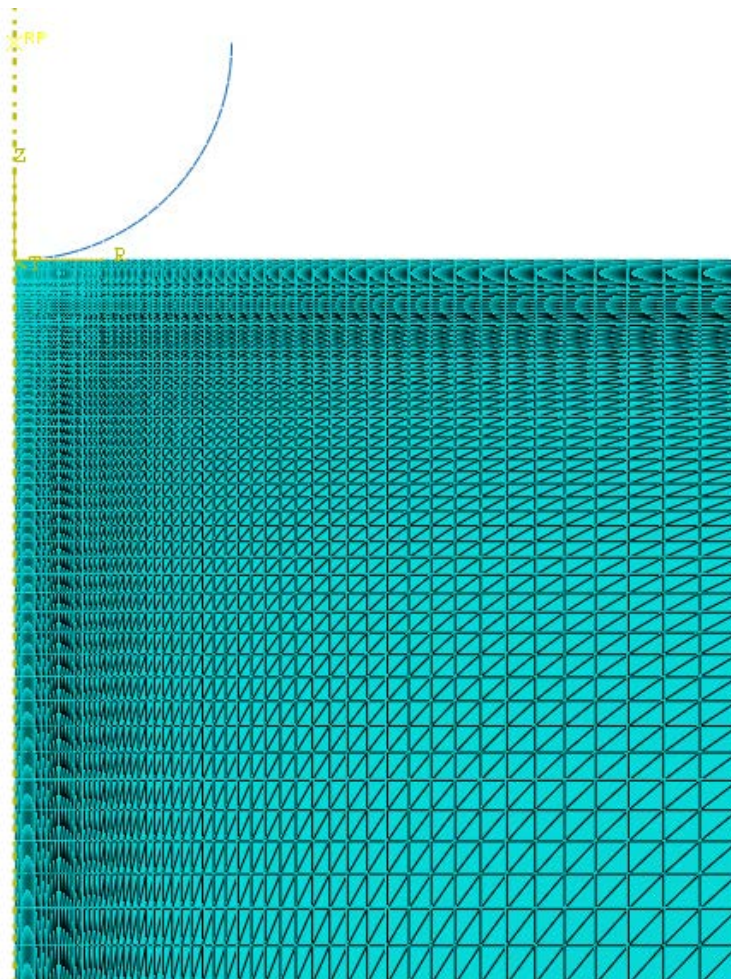
Figure 6: Finite element mesh

Element Types

The Mesh module can generate meshes containing the element shapes shown on page 22. Most elements in Abaqus/Standard and Abaqus/Explicit correspond to one of the shapes shown; that is, they are topologically equivalent to these shapes. For example, although the elements CPE4, CAX4R, and S4R are used for stress analysis, DC2D4 is used for heat transfer analysis, and AC2D4 is used for acoustic analysis, all five elements are topologically equivalent to a linear quadrilateral.

Every mesh region has one or more Abaqus element types assigned to it by default. Each element type corresponds to an element shape that can be used in the region. For example, a shell mesh region typically has a quadrilateral and a triangular element type assigned to it by default. However, you can change the element assignment for any ABAQUS element that is topologically equivalent to the element shape assigned to the region. As a result, you can choose to mesh a shell region with only all triangular elements, and ABAQUS/CAE ignores the quadrilateral element assignment.

To change the element assignment to an ABAQUS element that is topologically equivalent to the element shape assigned to the region, select **Mesh➞Element Type** from the main menu bar. Similarly, you can select **Mesh➞Controls** to select the element shape for meshing.

However, since no element type checking is done until you submit the analysis, it is possible to choose an element that is inappropriate for the analysis you will be conducting. For example, Abaqus/CAE does not prevent you from specifying heat transfer elements such as DC2D4, even though you may be conducting a stress analysis.

Job

Once you have finished all of the tasks involved in defining a model (such as defining the geometry of the model, assigning section properties, and defining contact), you can use the Job module to analyze your model. The Job module allows you to create a job, to submit it to Abaqus/Standard or Abaqus/Explicit for analysis, and to monitor its progress. If desired, you can create multiple models and jobs and run and monitor the jobs simultaneously.

In addition, you have the option of creating only the analysis input file for your model. This option allows you to view and edit the input file before submitting it for analysis. You can also view and edit the analysis keywords for a model by selecting **Model➞Edit Keywords ➞model name** from the main menu bar.

Visualisation

The Visualisation module provides graphical display of finite element models and results. It obtains model and result information from the output database; you can control what

information is placed in the output database by modifying output requests in the Step module. You can view your model and results by producing any of the following plots:

Undeformed shape

An undeformed shape plot displays the initial shape or the base state of your model.

Deformed shape

A deformed shape plot displays the shape of your model according to the values of a nodal variable such as displacement.

Contours

A contour plot displays the values of an analysis variable such as stress or strain at a specified step and frame of your analysis. The Visualisation module represents the values as customised colored lines, colored bands, or colored faces on your model.

Symbols

A symbol plot displays the magnitude and direction of a particular vector or tensor variable at a specified step and frame of your analysis. The Visualisation module represents the values as symbols (for example, arrows) at locations on your model.

Material orientations

A material orientation plot displays the material directions of elements in your model at a specified step and frame of your analysis. The Visualisation module represents the material directions as material orientation triads at the element integration points.

X–Y data

Cantilever Beam Bending Analysis

Type of Solver: ABAQUS CAE/Standard

TLP: Bending and Torsion of Beams -http://www.doitpoms.ac.uk/tlplib/beam_bending/index.php

Continuum Mechanics – Beam Bending

Problem Description:

Consider the cantilever beam shown below. The beam is made from aluminium, which has a Young's modulus of E = 70 GPa, a shear modulus of G = 25 GPa, and a Poisson's ratio of ν = 0.33. The beam is 1 m in length (L = 1) and has a square section with $a = b = 0.025$ m. When a transverse load is applied at some distance (x) along the beam length, a bending moment, *M*, is generated, where:

$$M = EI\frac{d^2y}{dx^2} = F(L-x) \tag{1}$$

The deflection of the beam is given by:

$$\delta = \frac{Fx^2(3L-x)}{6EI} \tag{2}$$

I is the second moment of area. For a square cross section:

$$I = \frac{a^4}{12} \tag{3}$$

X is the distance from the clamped end!!
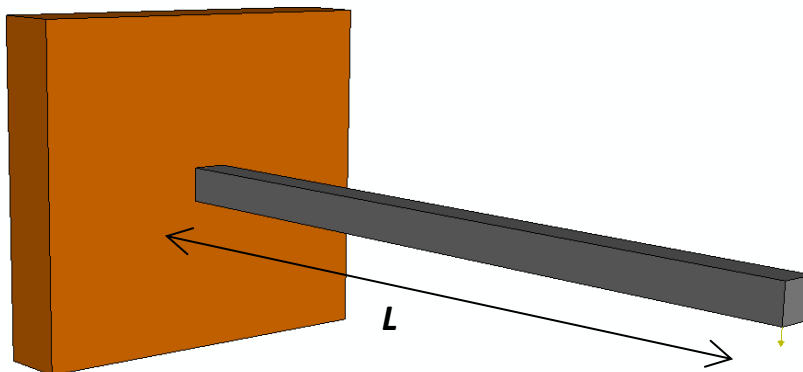
(a) Using a 1-dimensional finite element model, compute the deflection of a cantilever beam loaded at its end with a force of 80 N. Compare the FEM predicted deflections with those predicted by ordinary beam bending theory. Assume that the beam is made from aluminium, is homogenous and isotropic, and that it behaves in a linear elastic fashion.

(b) Using a 3-dimensional finite element model, compute the deflection of a cantilever beam loaded at its end with a force of 80 N. Compare the FEM predicted deflections, with those predicted by ordinary beam bending theory. Assume that the beam is made from aluminium, is homogenous and isotropic, and that it behaves in a linear elastic fashion.

(c) Using the 3-dimensional FE model, investigate the effect of **mesh density** on the predicted FEM deflections. Re-mesh the cantilever beam with 5000, C3D8R elements, and re-run the analysis. Compare the subsequent FEM predicted deflections with those of ordinary beam bending theory.

(d) Investigate the effect of **element type** on the predicted FEM deflections. Re-mesh the cantilever beam with 5000, C3D20R elements, and re-run the analysis. Compare the subsequent FEM predicted deflections with those of ordinary beam bending theory. Why might these elements be more accurate?

(e) Using the 3-D FE model (5000 C3D20R elements) plot the distribution of stress through the section of the model at $x = 0.1$.

(f) Compare the predicted stress at $x = 0.1$ for $y = 0.004$ and $y = 0.0125$ (where y is the distance from the neutral axis) with the stress at those positions predicted by ordinary beam bending theory. Plot these predictions on the same graph.

Solution (a):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database with Standard/Explicit Model.**

- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Cantilever_1D`

A. MODULE→PART

Under the Part module, we will construct the beam (1-D)

1. From the main menu bar, select **Part→Create**

2. The **Create Part** Dialogue box appears. Name the part `Cantilever Beam` and fill in the options as shown in **Fig.A1**. Click Continue to create the part.

3. From the main menu bar, select **Add→Line→Connected Line**

    (a) Select the co-ordinates (*0, 0)* for the first vertex (***enter***) as shown in **Fig.A2.**

    (b) Select the co-ordinates (*1, 0)* for the second vertex (***enter***).

    (c) Click **X** in the prompt area.

    (d) Click **Done** in the prompt area
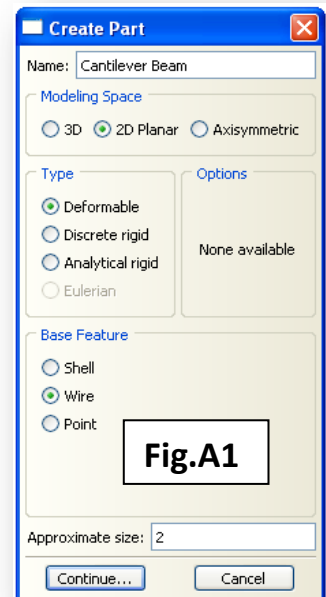
Fig.A1
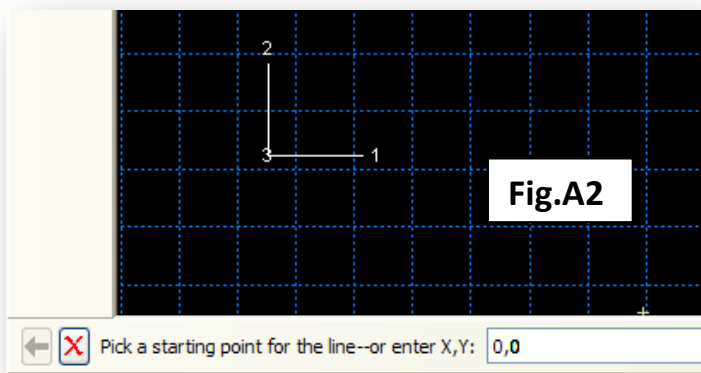
Fig.A2

The part, `Cantilever Beam` will now appear in the **Viewer** window, as a 1-dimensional beam. The following tasks must be completed:

- The beam section geometry must be defined.

- The beam material properties must be defined.

- The boundary conditions (constraints and loads) must be defined.

- A mesh must be assigned.

B. MODULE→PROPERTY

In this module (property), you will define the beam geometry (width and height), you will define the beam material properties (*E*, *G* and *v*) and you will assign these material properties to the beam.

1. From the main menu bar, select **Section→Create**. The **Create Section Dialogue** box will open as shown in **Fig.B1**. Name the section `Cantilever Section`.
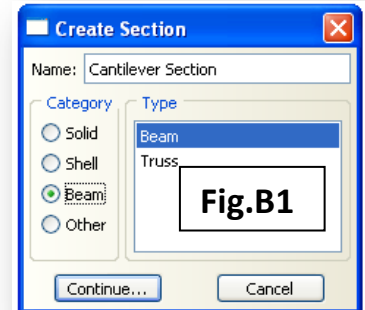
      (a) Under **Category**, choose **Beam**.

      (b) Under **Type**, choose **Beam**.

      (c) Click **Continue**.

Fig.B1

2. The **Edit Beam Section** dialogue box will open (**Fig.B2**). Next to **Profile Name**, click **Create** and the **Create Profile** dialogue box will open (**Fig.B3**). Name the profile `Cantilever Profile`. Select **Rectangular** and click **Continue**. The **Edit Profile** dialogue box will open (**Fig.B4**). Enter the cantilever cross sectional dimensions (**a = b = 0.025 m**) and click **OK**.

3. In the **Edit Beam Section** dialogue box, next to **Material name**, click **Create**, and the **Edit Material** Dialogue box will open (**Fig.B5**). Name the material `Aluminium` and then Click **Mechanical→Elasticity→Elastic.** Specify the elastic material properties, *E* and *v* and click **OK**. Specify the section Poisson's ratio to be 0.33 in the **Edit Beam Section** dialogue box. Click **OK**.

Fig.B2

Fig.B3

Fig.B4

Fig.B5

17
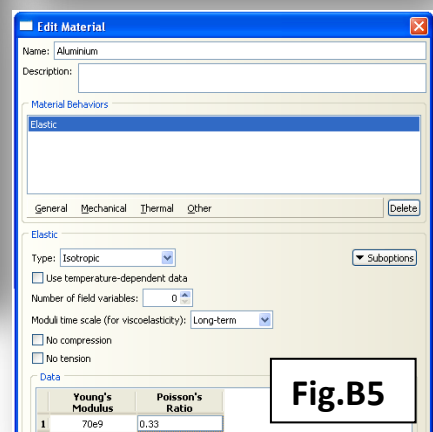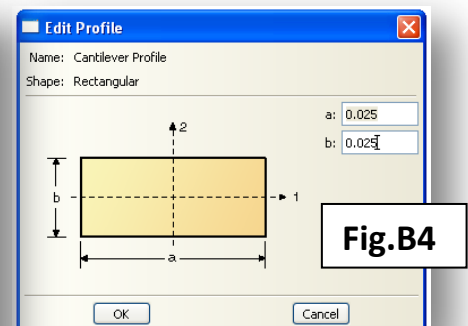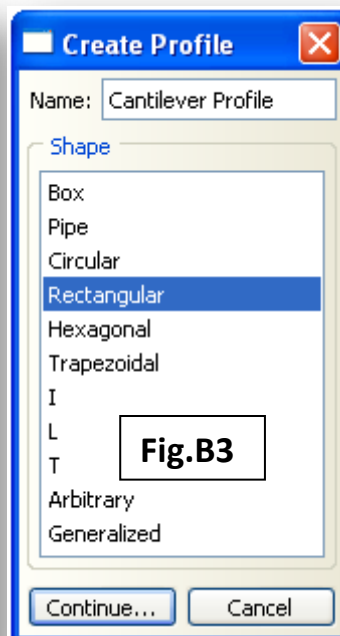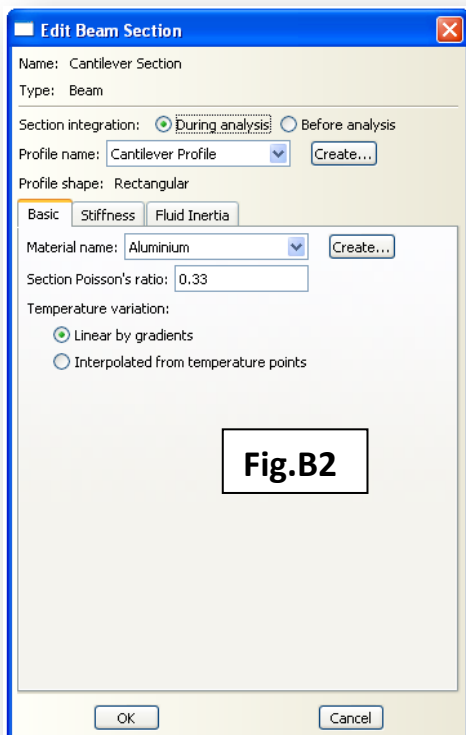
4.   From the main menu bar select **Assign→Section**. Use the mouse cursor to select the `Cantilever Beam` part and select **Done** in the prompt area. The **Edit Section Assignments** dialogue box will open as shown in **Fig.B6**.

> (a) Check that `Cantilever Section` is selected under the **Section** options.
>
> (b) Check that the **Type** is **Beam.**
>
> (c) Click **OK**.

Fig.B6

It is now necessary to define a beam orientation (*this is important for the second moment of area (*I*) calculation, particularly if the beam has geometry where a ≠ b. In this case, a = b, and the moment of area is independent of the loading direction; however, by default, ABAQUS requires a beam orientation to be defined for all beam sections).
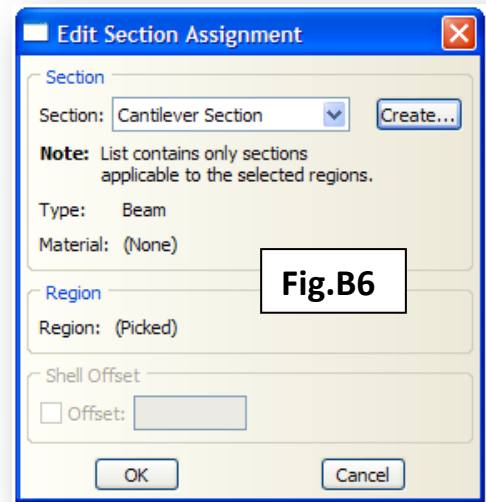
5.   From the main menu bar select **Assign→Assign Beam Section Orientation**. Use the mouse cursor to select the part in the **Viewer** and click **Done** in the prompt area. The default orientation can be selected by pressing **Enter**, clicking **OK** and then clicking **Done**.

C. MODULE→ASSEMBLY

In the assembly module, multiple parts can be 'assembled' into an assembly of parts. This is done by creating 'instances' of each part. In this case, we have only one part (`Cantilever Beam`). ABAQUS still requires, however, that an instance of this part is created. (FYI multiple instances of a single part can be created if required.)

Fig.C1

1.   From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open (**Fig.C1**).

> (a) Under Instance **Type**, select **Independent**.

D. MODULE→STEP

In the step module, you will define the type of analysis that is to be undertaken (static in this case).

1.   From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear (**Fig.D1**). Name the step `Load Step`.

Fig.D1

18

2. Select **General** from the **Procedure Type** options.

3. Select **Static, General** from the list of analysis types. Click **Continue** and **OK**.

E. MODULE→INTERACTION

There are no interactions in this analysis.

F. MODULE→LOAD

In the load module you define the boundary conditions (constraints and loads). You will constrain one end of the cantilever beam to be fixed (zero displacements) and you will define an 80 N load at the free end of the beam.

1. From the main menu bar, select **Load→Create**. The **Create Load** Dialogue box will open (**Fig.F1**).

> (a) Name the load `Concentrated Load`.
>
> (b) Choose `Load Step` as the **Step** option.
>
> (c) Choose **Mechanical** for the **Category**.
>
> (d) Choose **Concentrated Force** for the **Type**.
>
> (e) Click **Continue**.
>
> (f) Using the mouse cursor, select the second vertex (node)
>
> (g) Click **Done** in the prompt area.

Fig.F1
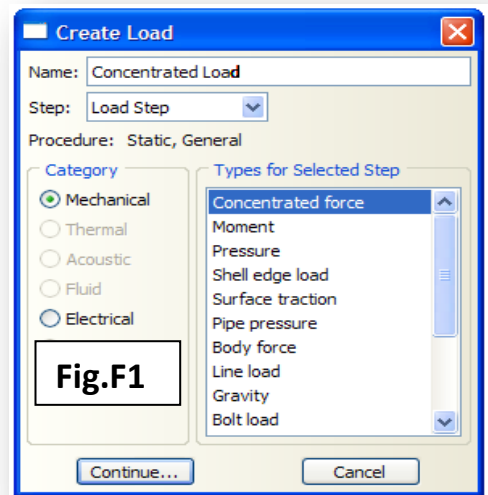
2. The **Edit Load** dialogue box will open, as shown in **Fig.F2**. You now need to specify a transverse load of 80 N.

> (a) Input CF1 = 0 (load in the x direction)
>
> (b) Input CF2 = -80 (load in the y direction)
>
> (c) Toggle off Follow Nodal Rotation

This ensures that the load is continuously applied in the y direction and not in a direction normal to the tangent of the node.

> (d) Click **OK**.

Fig.F2

19
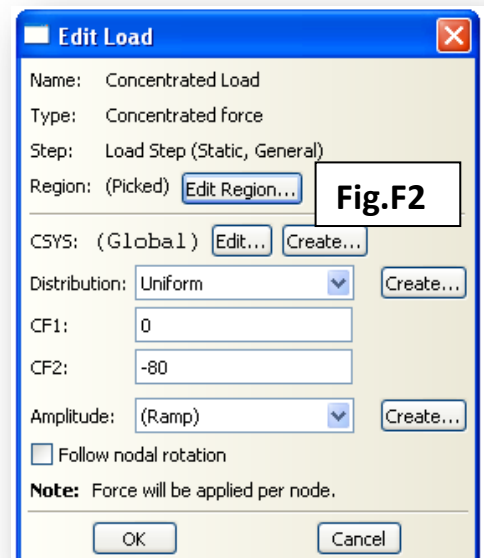
It is now necessary to define the constraint boundary conditions (i.e. to fix the opposite end of the beam so that it cannot move).

3. From the main menu bar, select **BC→Create**. The **Create Boundary Condition** dialogue box will appear, **Fig.F3**.

(a) Name the boundary condition, `Clamped End`.

(b) Choose `Load Step`, for the **Step** type.

(c) **Choose Mechanical** for the **Category**.

(d) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step option**.

(e) Click **Continue**.

(f) Use the mouse cursor to select the first vertex (node) which is going to be clamped.

(g) Click **Done** in the prompt area.
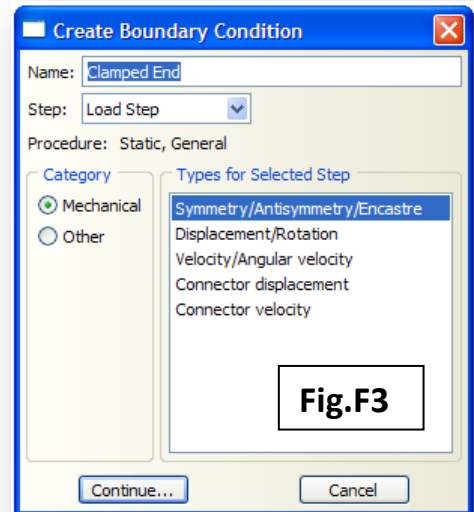
The Edit Boundary Condition dialogue box will open as seen in **Fig.F4**.

(a) Select **Encastre** as the boundary condition.

(b) Click **OK**.
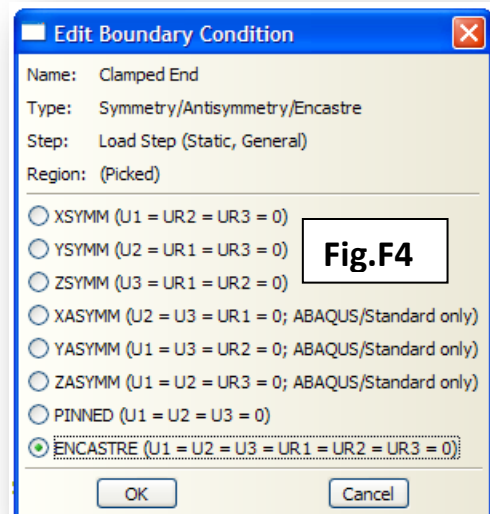
G. MODULE→MESH

In this mesh module, you will mesh the `Cantilever Beam` instance, by assigning seeds (nodal positions), mesh controls and element types.

1. From the main menu bar, select **Seed→Edges**.

2. Use the mouse cursor and select the `Cantilever Beam` instance and click **Done** in the prompt area.

In the **Local Seeds** dialogue box that appears, change the seeding **method** to **By Number**

3. Type **20** for the number of nodes along the beam length.

4. Click **OK**

5. From the main menu bar, select **Mesh→Element Type**. Using the cursor select the part instance and click **Done** in the prompt area. The **Element Type** dialogue box will appear as shown in **Fig.G1**.

6. Choose **Standard** from the **Element Library**.

7. Choose **Linear** for the **Geometric Order**.

8. Choose **Beam** for the **Family**.

9. Click **OK**.

10. From the main menu bar, select **Mesh→Instance**

11. Click **Yes** in the prompt area.

Fig.G1
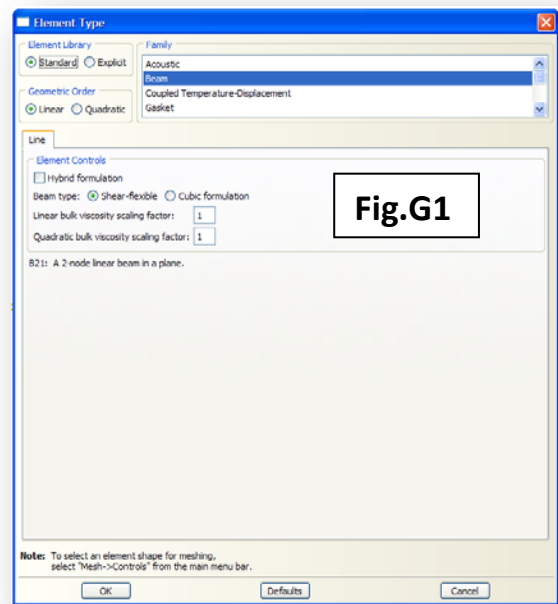
You have so far built the geometry, prescribed the beam section geometry, the beam material properties and the beam section orientation. You have created an instance of the `Cantilever Beam` part, defined a constraint boundary condition and a loading boundary condition. You have meshed the instance with 20, two-node, linear B21 beam elements. All that is now required is for the job to be submitted to the solver.

H. MODULE→JOB

In this module, you will submit the job to the solver for analysis.

1. From the main menu bar select **Job→Create**. The **Create Job** dialogue box will open.

    (a) Name the job `Cantilever_1D`.

    (b) Click **Continue**.

2. The **Edit Job** dialogue box will open:

    (a) In the **Description Field**, type `1D Cantilever Beam Bending`.

    (b) Click **OK**.

3. From the main menu bar, select **Job→Submit** and choose the `Cantilever_1D` job.

4. You can monitor the job progress by selecting **Job →Monitor** from the main menu bar. When the job is complete, you can view the results in the visualisation module.

I. MODULE→VISUALISATION

In this module you can view the results of your analysis, output xy data, operate on data and export images and movies.

1. From the main menu bar select **File→Open→Cantilever_1D.odb** (C:/Temp directory).

2. From the main menu bar select **Results→Field Output**. The **Field Output** dialogue box will appear as shown in **Fig.I1**.

> (a) From the **Output Variables** list select **U** (displacement)
>
> (b) Under **Component**, select **U2** (vertical displacements)
>
> (c) Click **OK**.

3. From the main menu bar select **Plot→Contours→Deformed Shape**.

Fig.I1
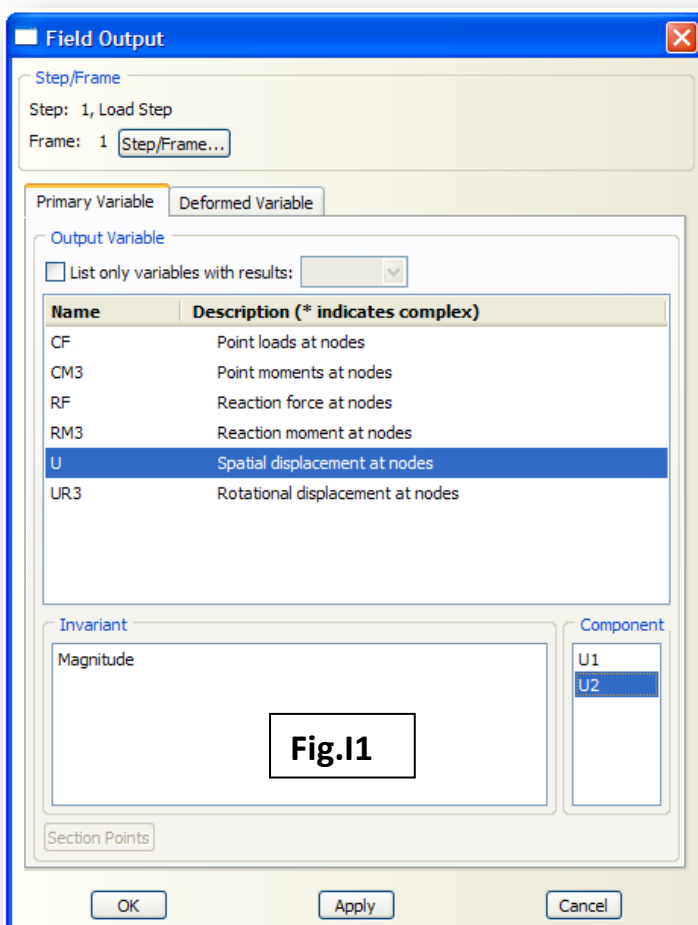
The deformed cantilever beam will now be displayed and coloured according to the contours of displacement. From the contours, you should be able to see that at the free end, the beam has deflected by 11.7 mm (**Fig.I3**).

```
U, U2
  +0.000e+00
  -9.755e-04
  -1.951e-03
  -2.926e-03
  -3.902e-03
  -4.877e-03
  -5.853e-03
  -6.828e-03
  -7.804e-03
  -8.779e-03
  -9.755e-03
  -1.073e-02
  -1.171e-02
```
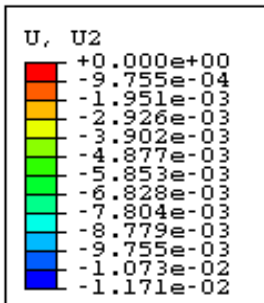
Fig.I3

Our interest now is in the full deflected profile. We'd like to compare the FEM predicted deflections with those from ordinary beam bending theory. To obtain the full FEM predicted profile:

4. From the main menu bar select **Plot→Contours→On Undeformed Shape**.

5.   From the main menu bar select **Tools→Path→Create**. The **Create Path** Dialogue box will open (Fig.I4).

> (a) Name the path, `Cantilever Path`.
>
> (b) Select **Edge List** as the **Type**.
>
> (c) Click **Continue**.

Create Path

Name: Cantilever Path

Type
- ○ Node list
- ○ Point list
- ● Edge list
- ○ Circular

Fig.I4

Continue...   Cancel   Tip...
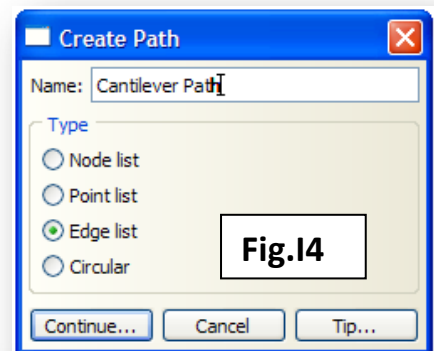
The **Edit Edge List Path** dialogue box will open (Fig.I5)

> (a) Select **Add Before** from the **Viewport Selection** options.
>
> (b) In the prompt area, choose **Feature Edge** (**Fig.I6**)
>
> (c) Using the mouse cursor, select the first element of the beam in the **Viewer**.

Edit Edge List Path

Name: Cantilever Path

Type: Edge List

Path Definition

	Part Instance	Edge Labels
1	CANTILEVER BEAM-1	

Fig.I5

Viewport selections: Add Before... Add After...

OK   Cancel

Select edges to be inserted into the path | by feature edge

individually
by feature edge
by shortest distance
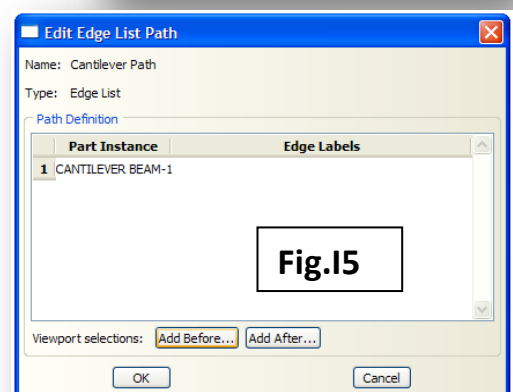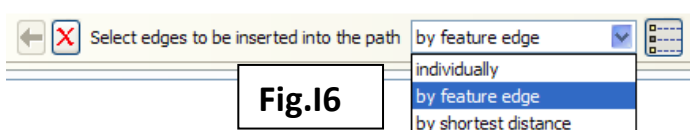
Fig.I6

> (d) Click **Done** in the prompt area and **OK** in the **Edit Edge List Path** dialogue box.

The nodes that form the path will be identified as shown in **Fig.I7**.

Fig.I7

Start: 1 2   3   4   5   6   7   8   9   10   11   12   13   14   15   16   17   18   19   20   End: 21

6. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path** from the list of options and click **Continue**. The **XY Data from Path** dialogue box will open (**Fig.I8**).

> (a) Select `Cantilever Path` from the **Path** options
>
> (b) Select **Deformed** from the **Model Plot** options.
>
> (c) Select **True Distance** as shown.
>
> (d) Clicking the **Field Output** icon will bring up the **Field Output** dialogue box (**Fig.I9**).
>
> (e) Select **U** as the primary variable and **U2** as the **Component**. Click **OK**.
>
> (f) To view the path plot, click **Plot**.
>
> (g) Save the data by clicking **Save As…**, and name the data **Al_Deflection_Elastic_1D**.

Fig.I8

Fig.I9
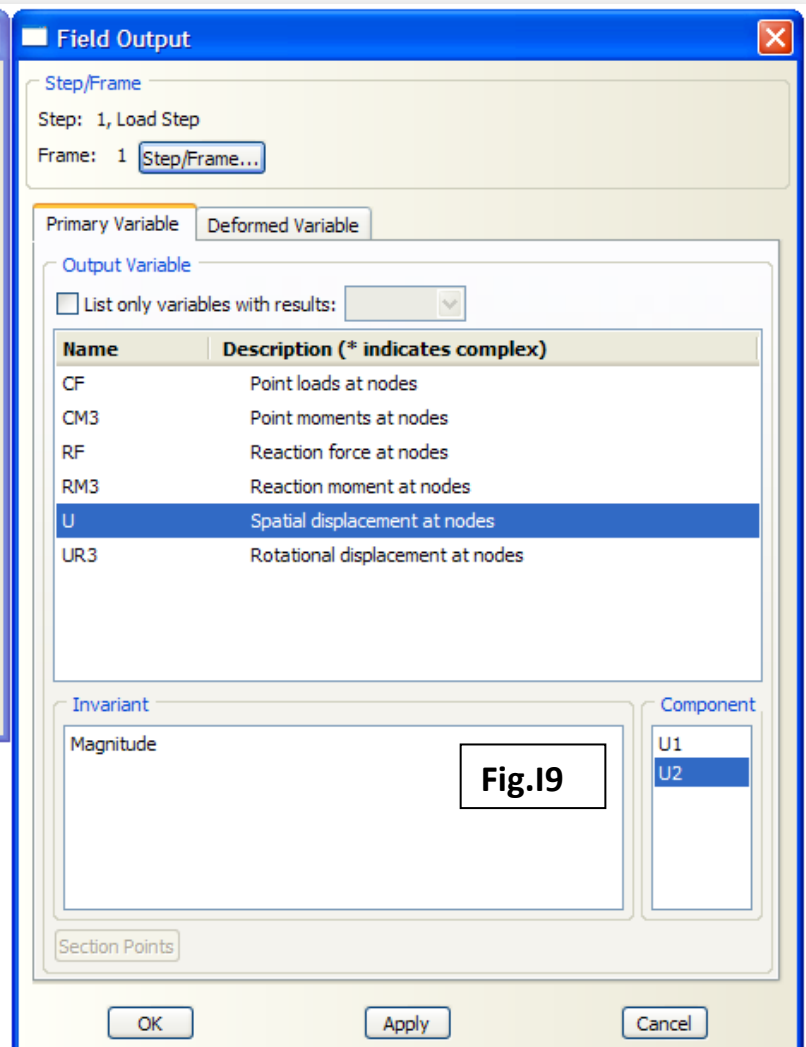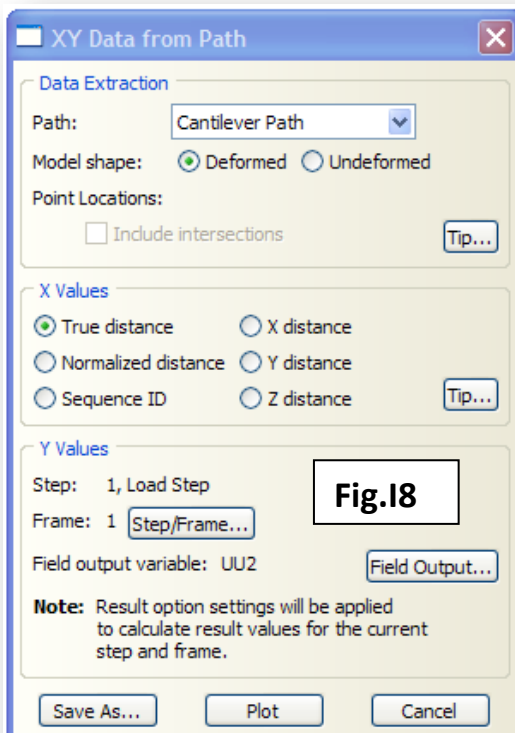
7. To access the data so that you can compare the predictions to those of ordinary beam bending theory, select **Tools→XY Data→Edit→Al_Deflection_Elastic_1D**. Copy and paste the data into Excel. The results should look as shown in **Fig.I10**.

$$\delta = \frac{Fx^2(3L-x)}{6EI}$$
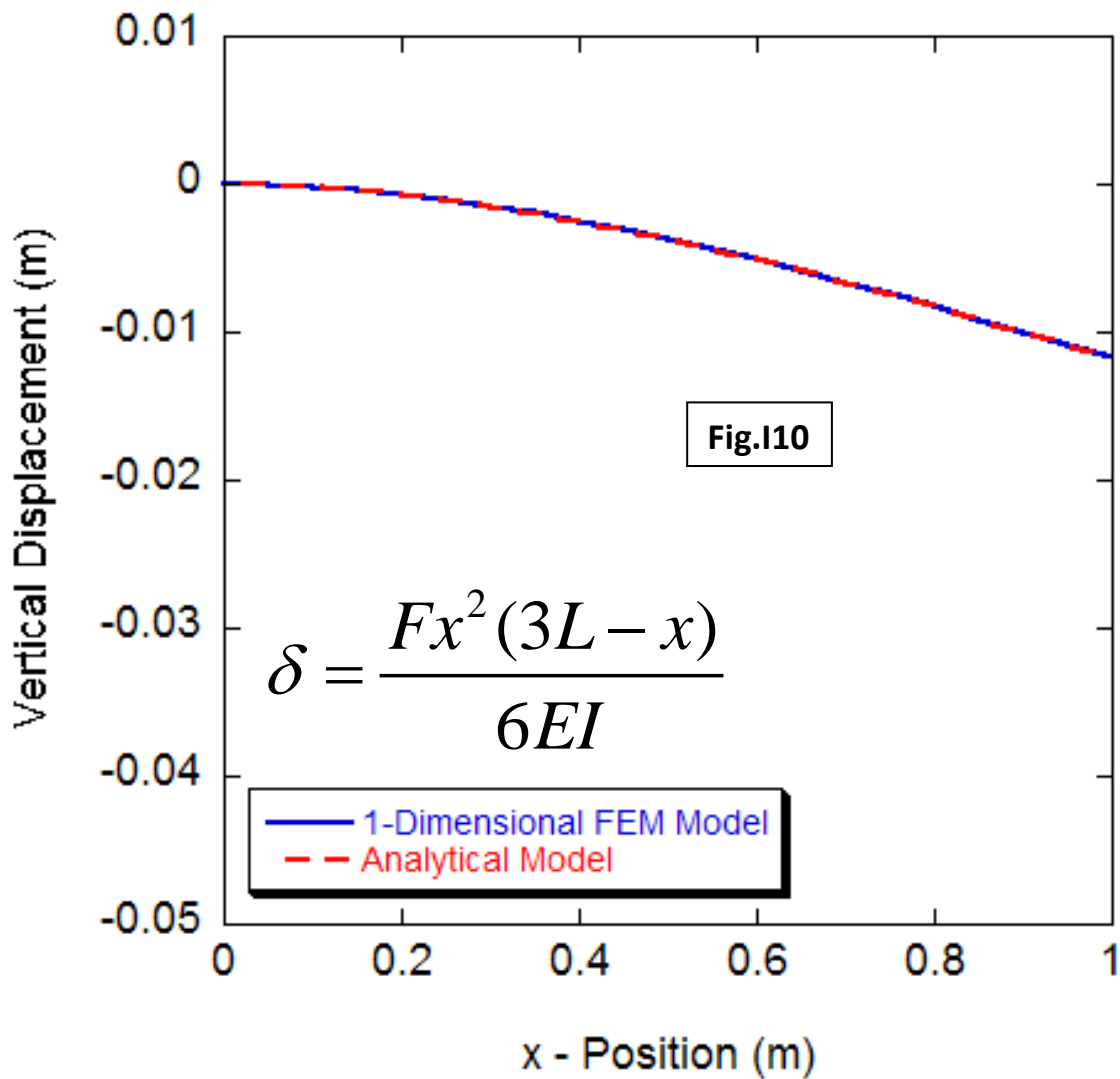
Solution (b):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database.**
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Cantilever_3D`.

A. MODULE→PART

Under the Part module, we will construct the beam (3-D)

1. From the main menu bar, select **Part→Create**

2. The **Create Part** Dialogue box appears. Name the part `Cantilever Beam` and fill in the options as shown in **Fig.A1**. Click Continue to create the part.

3. From the main menu bar, select **Add→Line→Rectangle**

    (a) Select the co-ordinates (*0, 0*) for the first vertex (*enter*) as shown in **Fig.A2.**

    (b) Select the co-ordinates (*1, 0.025)* for the second vertex (*enter*).

    (c) Click ☒ in the prompt area.

    (d) Click **Done** in the prompt area.

Fig.A2

Fig.A1
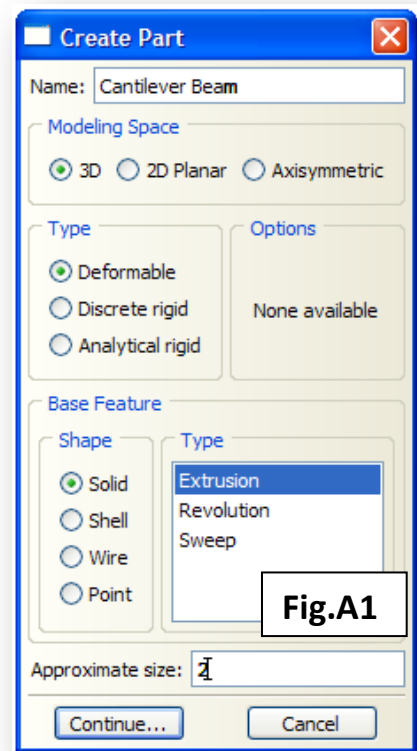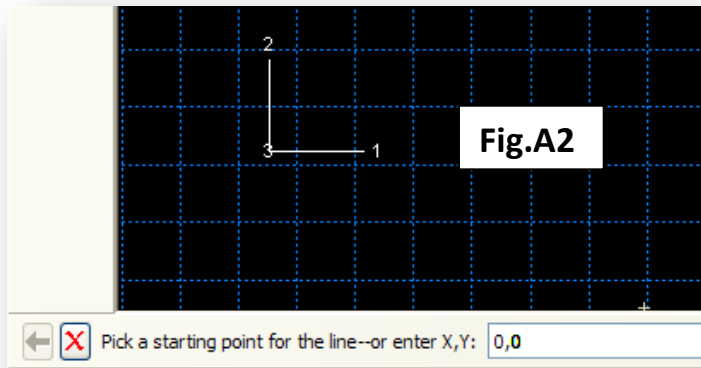
4. The **Edit Base Extrusion** dialogue box will open. In the **Depth Field**, type **0.025** and click **OK**. The part, `Cantilever Beam` will now appear in the **Viewer** window, as a 3-dimensional beam. The following tasks must be completed:

- The beam material properties must be defined.
- The boundary conditions (constraints and loads) must be defined.
- A mesh must be assigned.

B. MODULE→PROPERTY

In this module (property), you will define the beam material properties (*E* and *ν*) and you will assign these material properties to the beam.

1. From the main menu bar, select **Section→Create**. The **Create Section Dialogue** box will open as shown in **Fig.B1**. Name the section `Cantilever Section`.

     (a) Under **Category**, choose **Solid**.

     (b) Under **Type**, choose **Homogenous**.

     (c) Click **Continue**.

Fig.B1
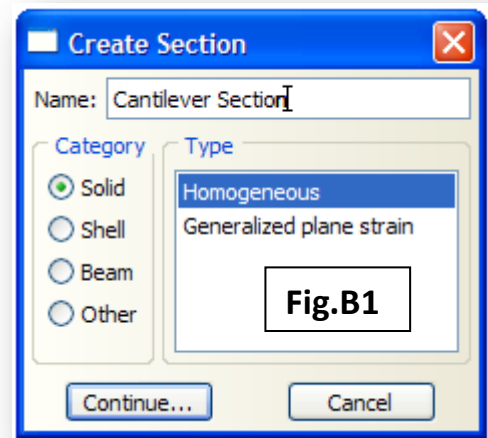
2. The **Edit Section** dialogue box will open (**Fig.B2**). Click **Create** and the **Create Material** dialogue box will open (**Fig.B3**). Name the material Aluminium. Enter a Young's Modulus of ***E* = 70 GPa**, and a Poisson's ratio of *ν* **= 0.33** and click **OK**.

Fig.B2

Fig.B3
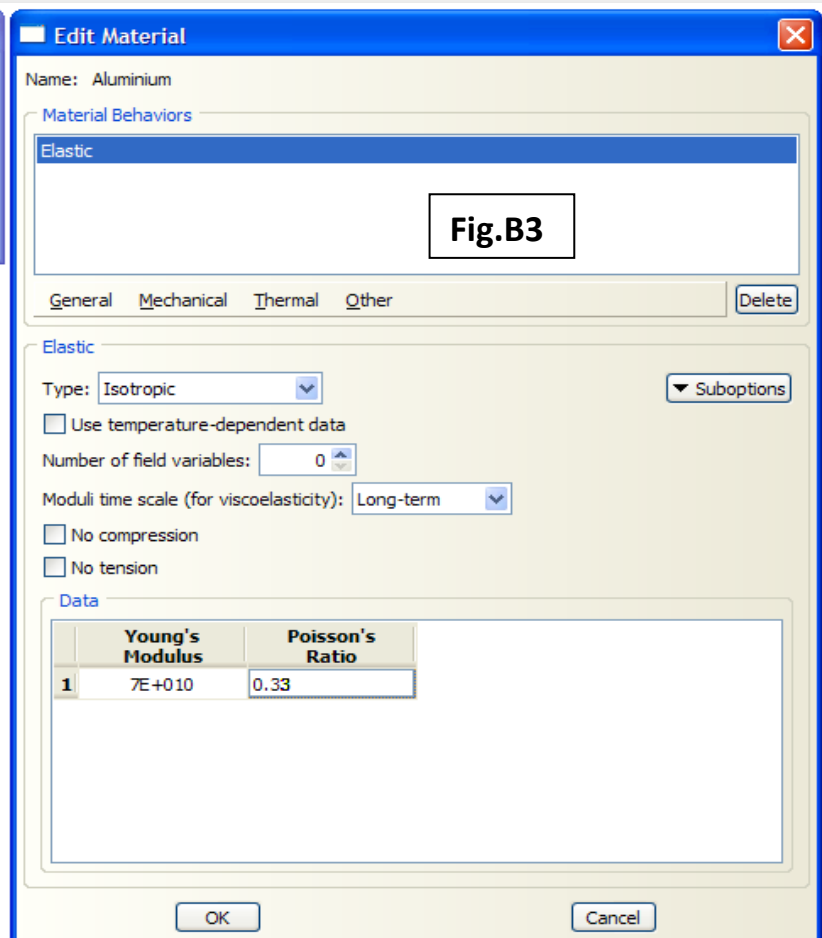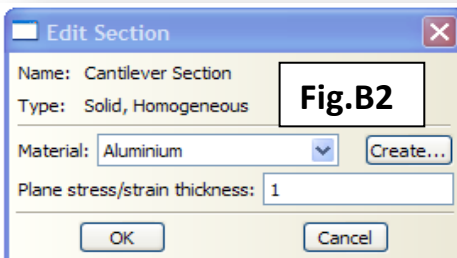
You now need to assign the `Cantilever Section` and the `Cantilever Material` to the 3-D beam part that you have created.

3. From the main menu bar select **Assign→Section**. Using the mouse cursor, select the part in the **Viewer** and click **Done**. The **Edit Section Assignment** dialogue box will open. Check that the **Section** that is chosen is the `Cantilever Section` that you created (it should be there by default!). Click **OK**. The part will change colour, which is an acknowledgement that that section has been assigned to the material.

C. MODULE→ASSEMBLY

In the assembly module, multiple parts can be 'assembled' into an assembly of parts. This is done by creating 'instances' of each part. In this case, we have only one part (`Cantilever Beam`). ABAQUS still requires, however, that an instance of this part is created. (FYI multiple instances of a single part can be created if required.)

1. From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open (**Fig.C1**).

    (a) Under Instance Type, select **Independent**.

    (b) Click **OK**.
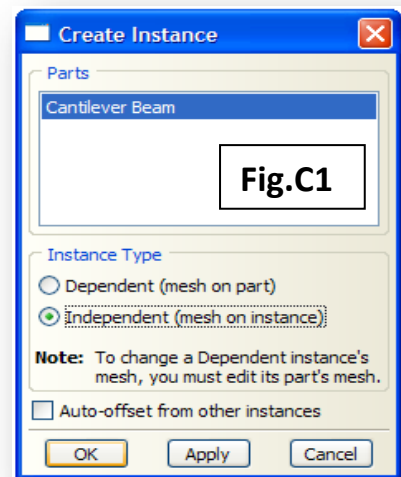
Fig.C1

D. MODULE→STEP

In the step module, you will define the type of analysis that is to be undertaken (static in this case).

1. From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear (**Fig.D1**). Name the step `Load Step`.

2. Select **General** from the **Procedure Type** options.

3. Select **Static, General** from the list of analysis types. Click **Continue**.

Fig.D1

E. MODULE→INTERACTION

There are no interactions in this analysis.

F. MODULE→LOAD

In the load module you define the boundary conditions (constraints and loads). You will constrain one end of the cantilever beam to be fixed (zero displacements) and you will define an 80 N load at the free end of the beam.

1. From the main menu bar, select **Load→Create**. The **Create Load** Dialogue box will open (**Fig.F1**).

    (a) Name the load `Concentrated Load`.

    (b) Choose `Load Step` as the **Step** option.

    (c) Choose **Mechanical** for the **Category**.

    (d) Choose **Concentrated Force** for the **Type**.

    (e) Click **Continue**.

    (f) Using the mouse cursor, select the two nodes as shown in **Fig.F2**.

    (g) Click **Done** in the prompt area and the **Edit Load** dialogue box will open (**Fig.F3**).

    (h) Type **0** for **CF1** and **CF3**. Type **-40** for **CF2** (i.e. 40 N load at each node).
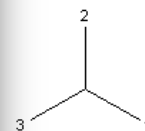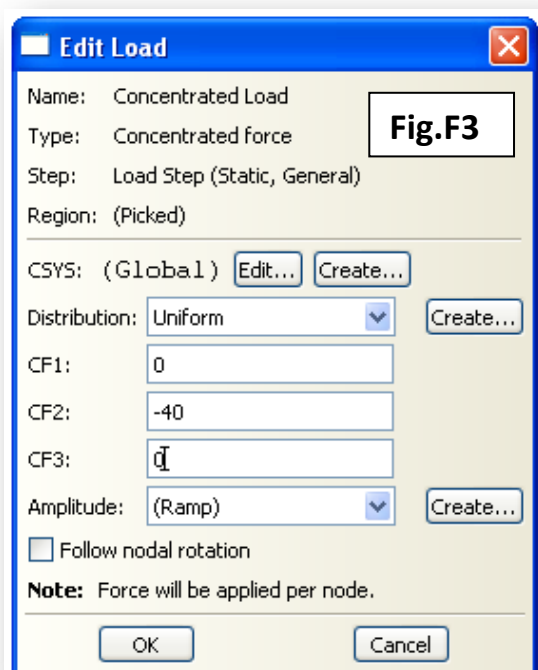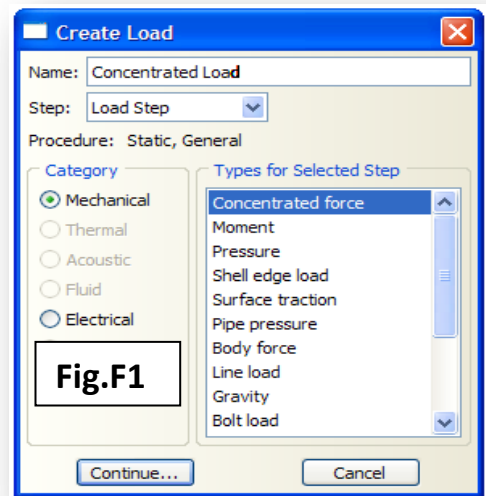
Fig.F1

Fig.F3

Fig.F2

    (i) Toggle off **Follow Nodal Rotation**.

    (j) Click **OK**.

2. From the main menu bar select **BC→Create**. The **Create Boundary Condition** dialogue box will open as shown in **Fig.F4**.

> (a) Name the boundary condition `Clamped End`.
>
> (b) Choose **Load Step** for the step.
>
> (c) Choose **Mechanical** for the **Category**.
>
> (d) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step**.
>
> (e) Click **Continue**.
>
> (f) Using the mouse cursor, select the opposite end of the bar to which the loading condition was specified (**Fig.F5**). The selected face will turn purple/red.
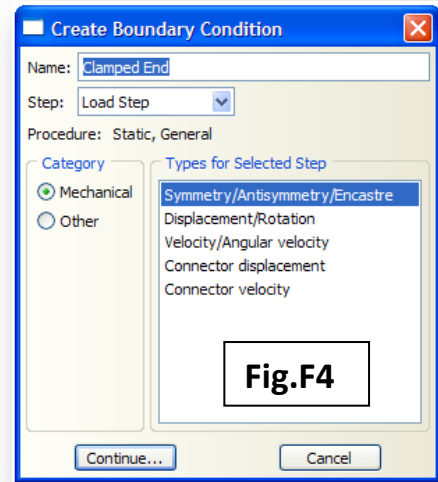
3. The Edit Boundary Condition dialogue box will appear.

> (a) Choose **Encastre** as the boundary condition.
>
> (b) Click **OK**.

G. MODULE→MESH

---
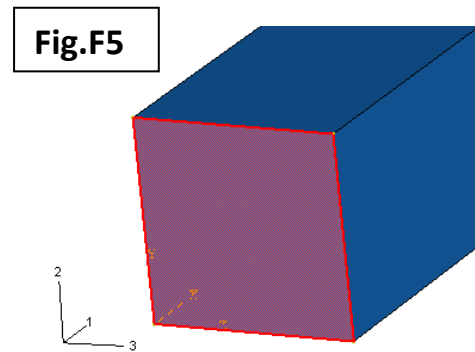
1. From the main menu bar select Mesh→Controls. The Mesh Controls dialogue box will appear as shown in **Fig.G1**.

> (a) Choose **Hex** as the **Element Shape**.
>
> (b) Select **Structured** for the **Technique**.
>
> (c) Click **OK**.
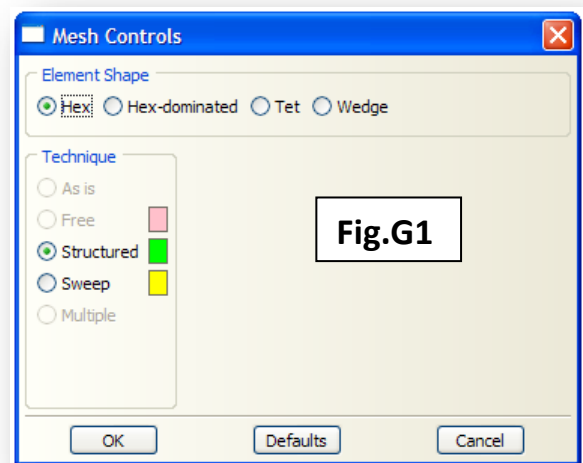
2. From the main menu bar select **Seed→Seed Edge by Number**. Using the mouse cursor (holding shift), select the end faces. Click **Done** in the prompt area. Type **2** for the number of elements along those face edges. Click **Done** in the prompt area. Repeat the operation for the side edges, choosing 80 seeds along each edge as seen in **Fig.G2**.

.
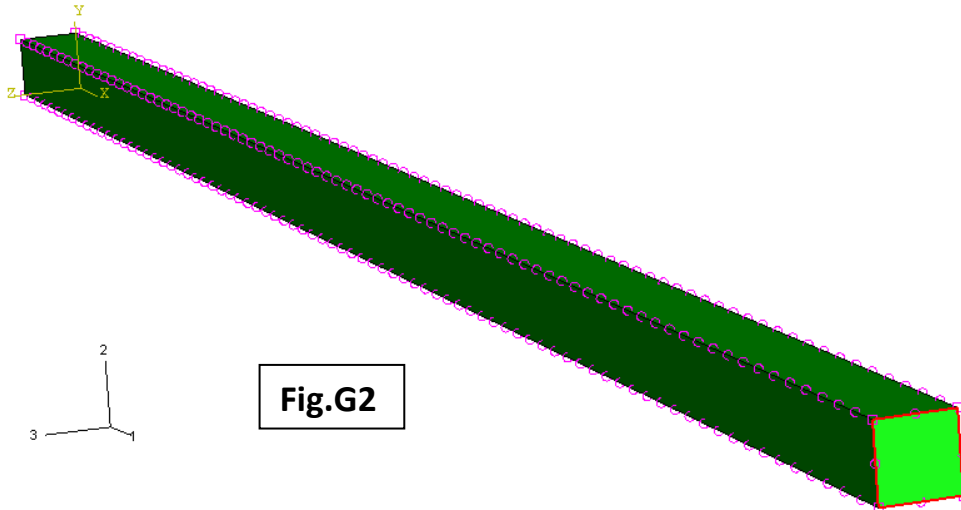
Fig.G2

3. From the main menu bar select Mesh→Element Type. The Element Type dialogue box will appear.

   (a) Choose **Standard** from the **Element Library**.

   (b) Choose **Linear** for the **Geometric Order**.

   (c) Choose **3D Stress** for the **Family**.

   (d) Click **OK** to choose **C3D8R** elements.

4. From the main menu select **Mesh→Instance**.

5. Click **Yes** in the prompt area to generate 320 elements on your part instance (**Fig.G3**).
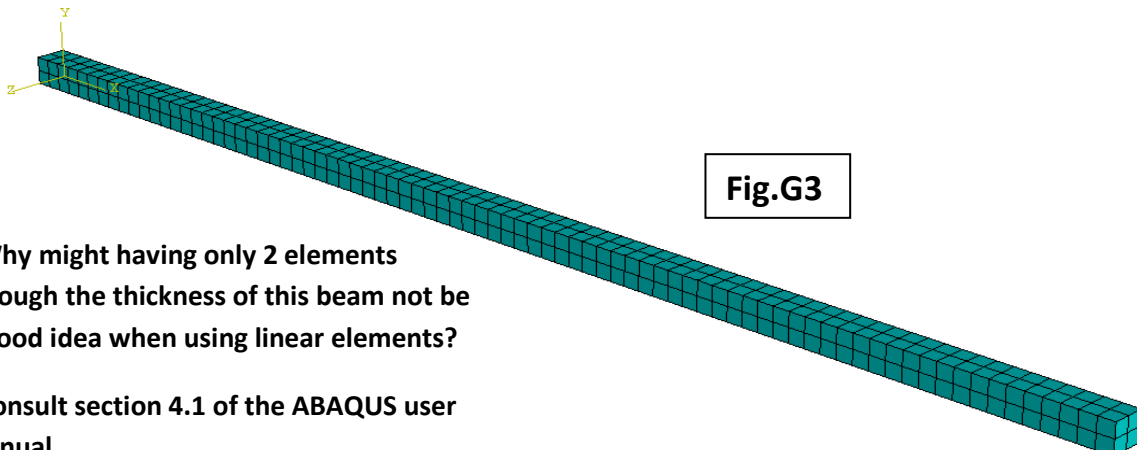
Fig.G3

*Why might having only 2 elements through the thickness of this beam not be a good idea when using linear elements?

*Consult section 4.1 of the ABAQUS user manual

H. MODULE→JOB

1. From the main menu bar select **Job→Create**. The **Create Job** dialogue box will open as seen in **Fig.H1**.

> (a) Name the job `Cantilever_3D`.
>
> (b) Select the `Cantilever_3D` Model.
>
> (c) Click **Continue**.

2. The Edit Job dialogue box will open.

> (a) In the **Description** field, type **3D Cantilever Bending Analysis**.
>
> (b) Click **OK**.

Fig.H1
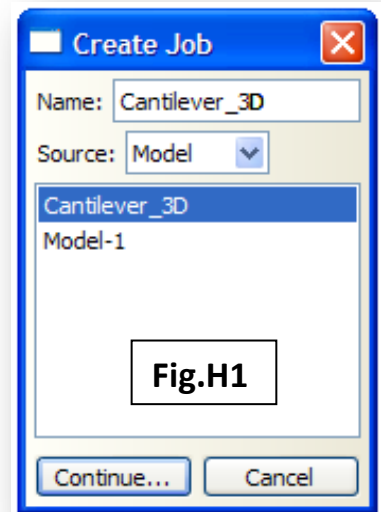
3. From the main menu bar **select Job→Submit→`Cantilever_3D`**. You can monitor the progress of your job by selecting **Job→Job Manager** from the main menu bar and **Monitor** from the **Job Manager** dialogue box. The analysis will take approximately 30 seconds (although this will depend on the CPU spec.). By default, the results will be saved in C:\Temp.

I. MODULE→VISUALISATION

1. From the main menu bar select **View→Toolbars→Views**. Choose plane 1-2 from the **Views** dialogue box (**Fig.I1**).

Fig.I1
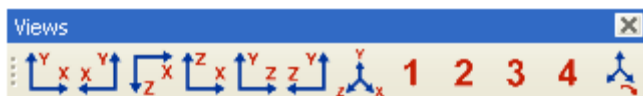
2. From the main menu bar select **Results→Field Output.** The **Field Output** options box will open. From the list of options, select **U** from the primary variables and **U2** as the component (**Fig.I2**).

> (a) Click **OK**.
>
> (b) Select **Contour** from the **Select Plot State** dialogue box and Click **OK**.
>
> (c) The deformed cantilever beam will be displayed.

32
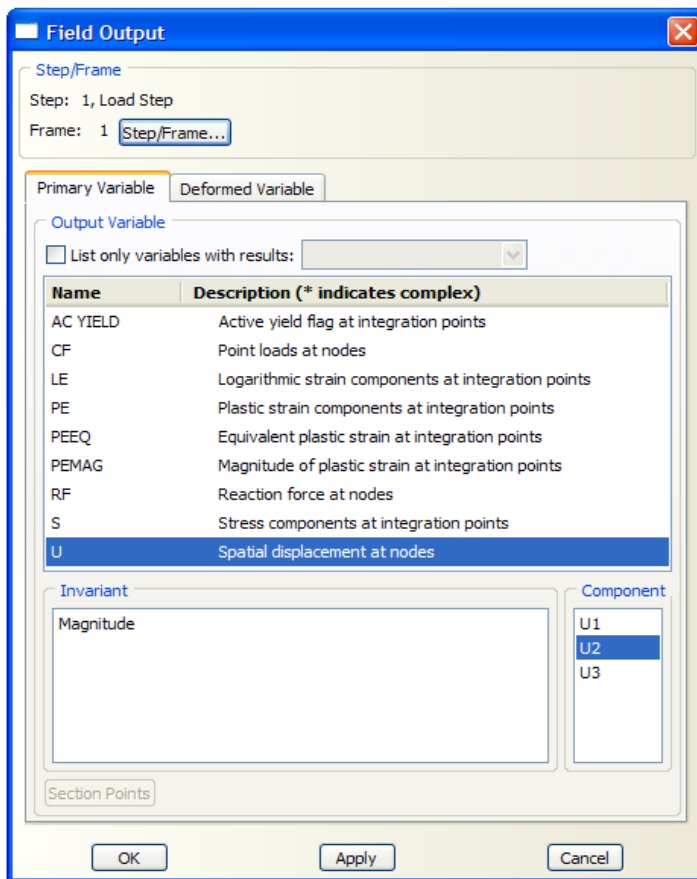
Fig.I2
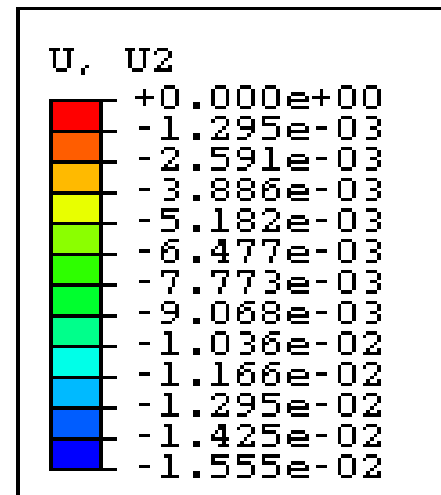
Our interest now is in the full deflected profile. To obtain this information:

3. From the main menu bar select **Plot→Contours→On Undeformed Shape**.

4. From the main menu bar select Tools→Path→Create. The Create Path dialogue box will open.

   (a) Choose **Edge list** as the **Type** (**Fig.I3**)

   (b) Name the path **Cantilever Path**.

   (c) Click **Continue**.

5. The Edit Edge List Path dialogue box will open.

   (a) Select **Add After** and **OK**.

   (b) In the prompt area select **By Feature Edge**.
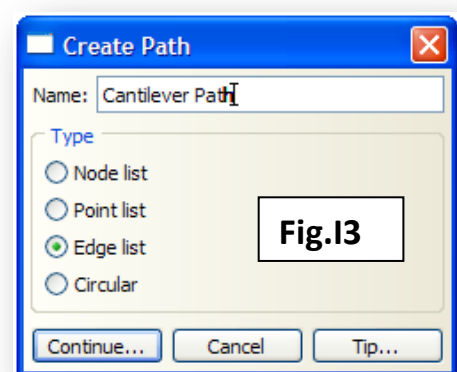
Fig.I3

6. Using the mouse cursor, select the first element edge.

      (a) Click **Flip** and then Click **Done** in the prompt area.

7. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path** (**Fig.I4**) and click **Continue**.
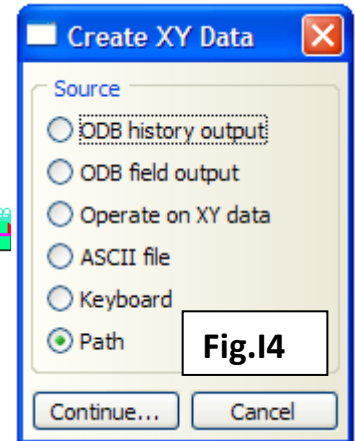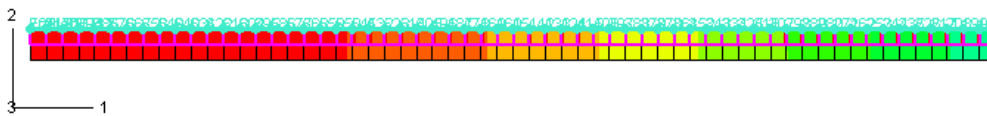
8. The **XY Data from Path** dialogue box will open (**Fig.I5**).

      (a) Select Cantilever Path from the **Path** options.

      (b) Select **Deformed** from the **Model Shape** options.

      (c) Select **True Distance** from the **X Values** option.

9. Select the **Field Output** icon and the **Field Output** dialogue box will open (**Fig.I6**).

      (a) Select **U** as the primary variable and **U2** as the component.

      (b) Click **OK** and then click **Plot** in the **XY Data from Path** dialogue box.

      (c) Click **Save As…** and name the file Al_Deflection_Elastic_3D. Click **OK**.

10. To access the data, from the main menu bar select :

XY Data→Edit→Al_Deflection_Elastic_3D.

Compare these predictions with those from ordinary beam bending theory.

Fig.I5

Fig.I6
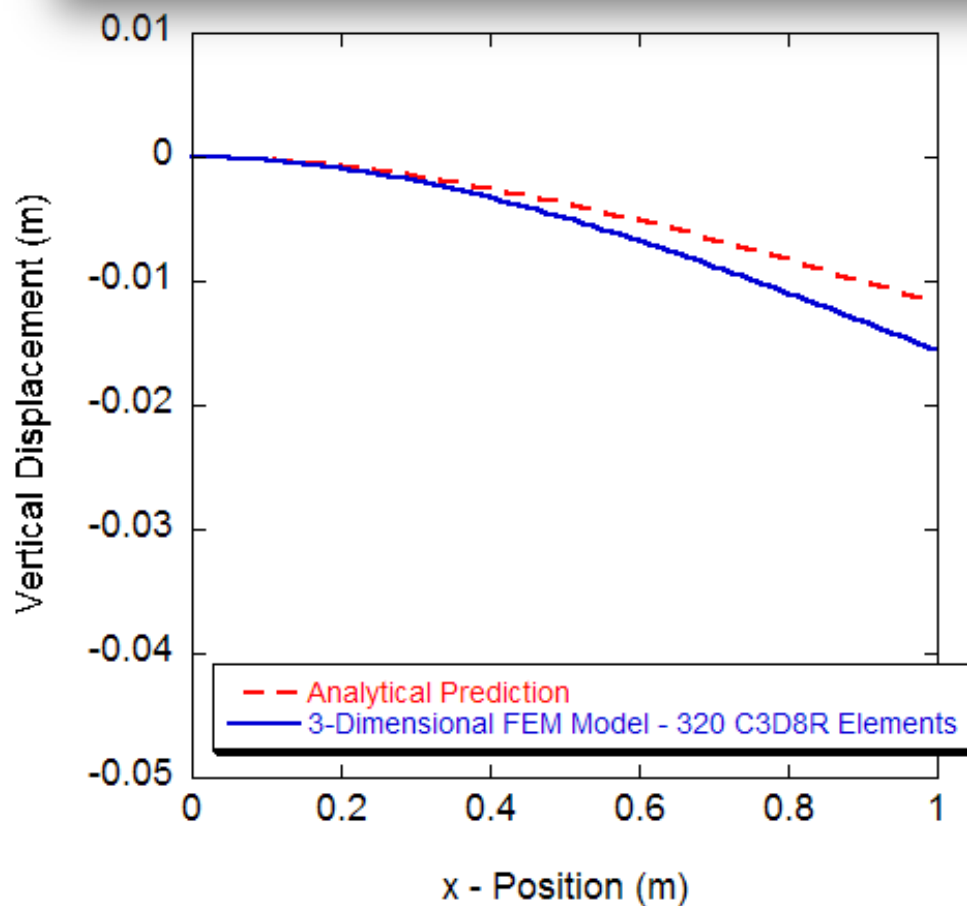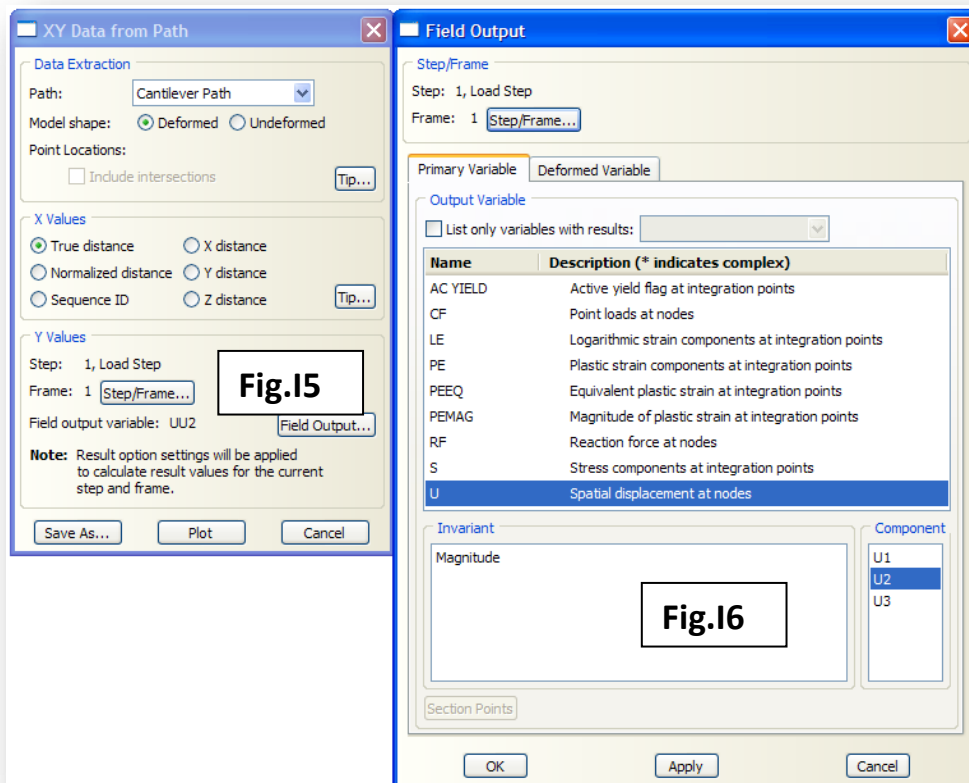
11. Note that in this plot, the FEM predicted deflection at the end of the bar is 15.5 mm. Ordinary beam bending theory predicts the deflection to be 11.7 mm. There is, therefore an apparent error of 24.5%. Why might this be?

Solution (c):

- Return to the **Mesh** module and re-seed the 3-D beam (200 seeds along the long edges and 5 seeds along the short edges).

- Re-mesh the beam; **Mesh→Instance**.

- Re-run the analysis; **Job→Job Manager→Submit**.

- Obtain the deflected profile; **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path……..**

- Compare the FEM predictions with those from beam bending theory.
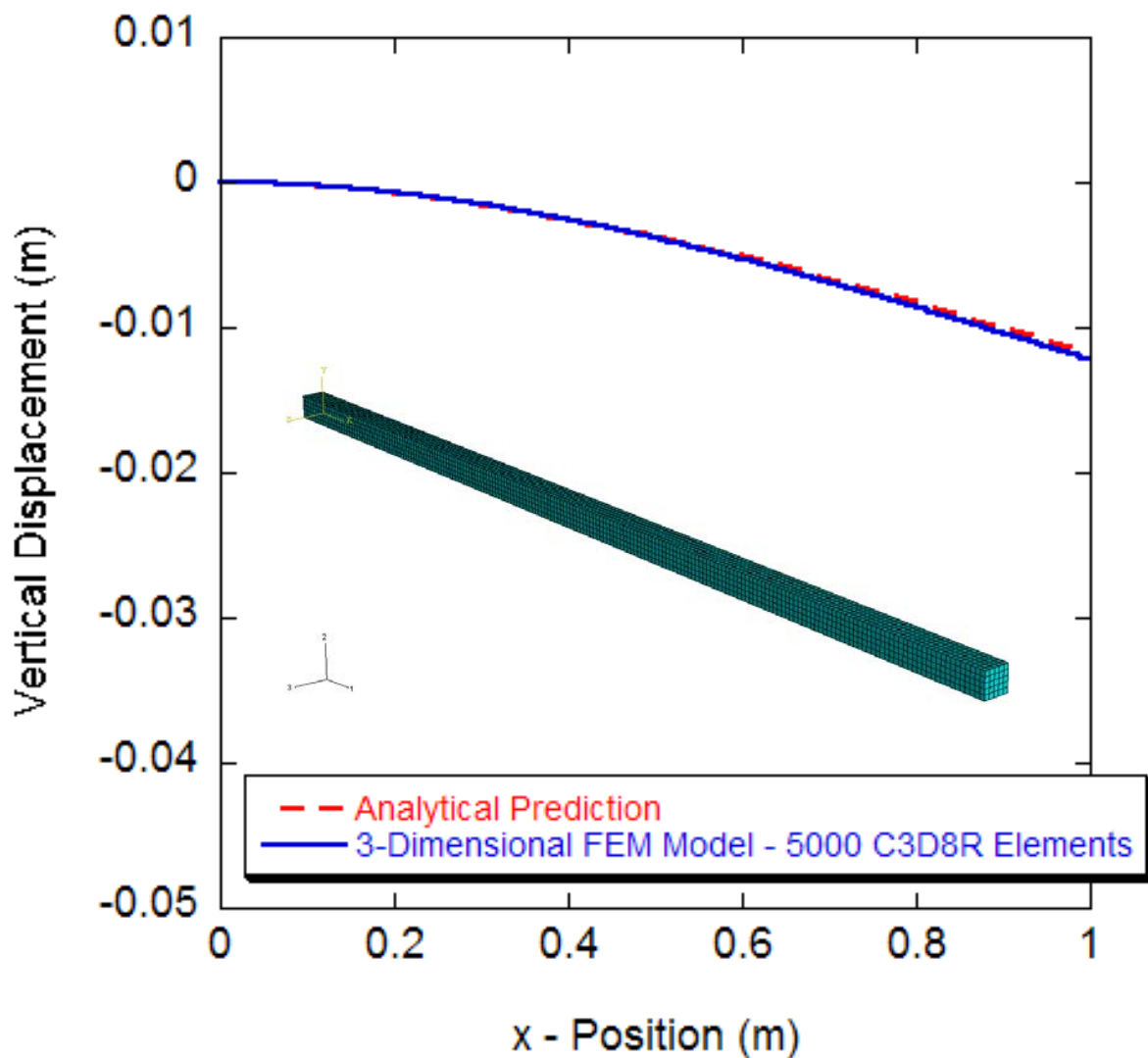
It is clear to see from these two analyses that the mesh density is important. However, there still remains a small discrepancy between the FEM predicted deflection at the end of the cantilever beam and that predicted by ordinary beam bending theory (7.8%).

Solution (d):

- Return to the **Mesh** module and change the element type; **Mesh→Element Type**. Select **Quadratic** from the **Geometric Order**. Choose **C3D20R** elements.

- Re-mesh the beam; **Mesh→Instance**.

- Re-run the analysis; **Job→Job Manager→Submit**.

- Obtain the deflected profile; **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open. Select **Path……..**

- Compare the FEM predictions with those from beam bending theory.
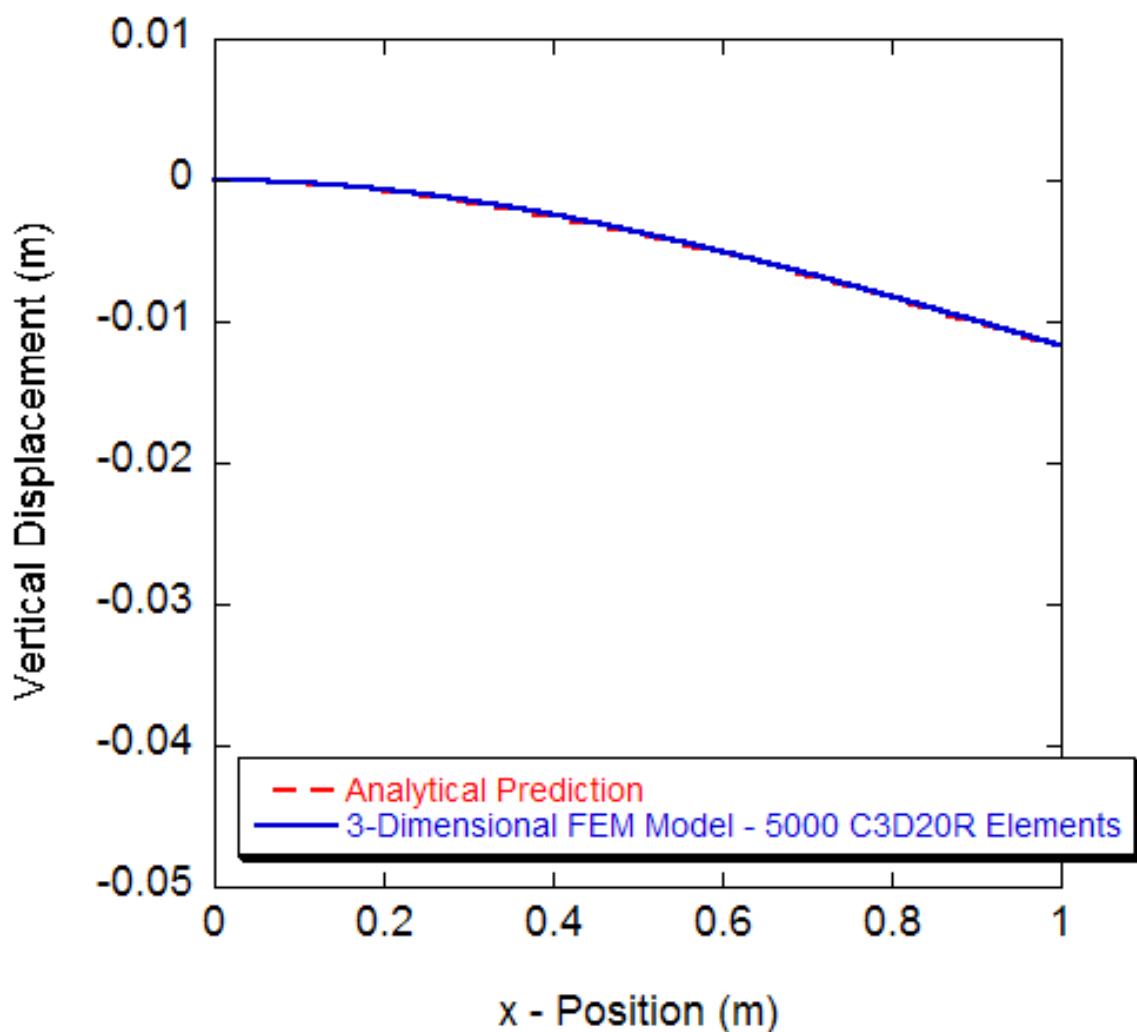
The error now falls to 0.17%.

Solution (e)

- Return to the **Mesh** module and change the element type; **Mesh→Element Type**. Select **Quadratic** from the **Geometric Order**. Choose **C3D20R** elements.

A. MODULE→VISUALISATION

---

1. From the main menu bar select **Results→Field Output**. The **Field Output** dialogue box will open (**Fig.A1**).

    (a) Select **S** as the primary variable.

    (b) Select **S11** as the stress component (axial stress)

    (c) Click **OK**.

2. The stress distribution will appear as shown in Fig.A2. There is a tensile state of axial stress at the top of the beam and a compressive axial stress state at the bottom. The peak tensile and compressive stresses are ~41 MPa .

Fig.A1
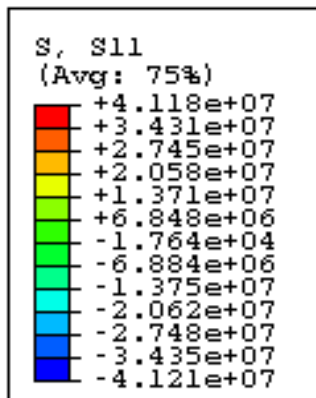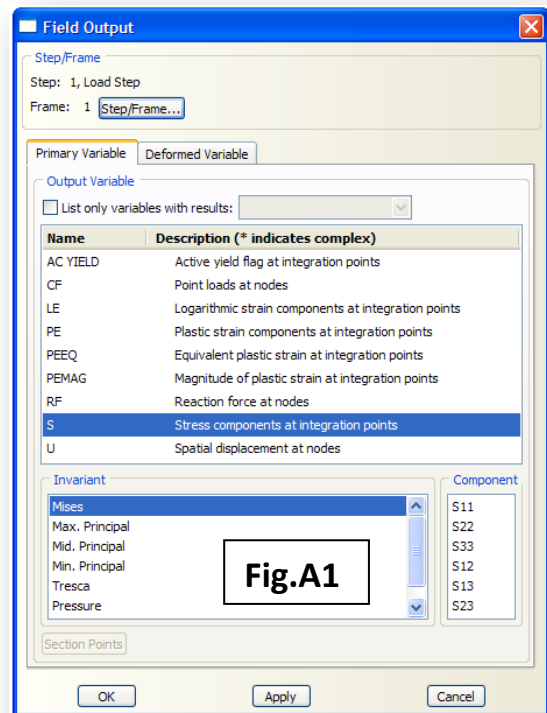
Fig.A2
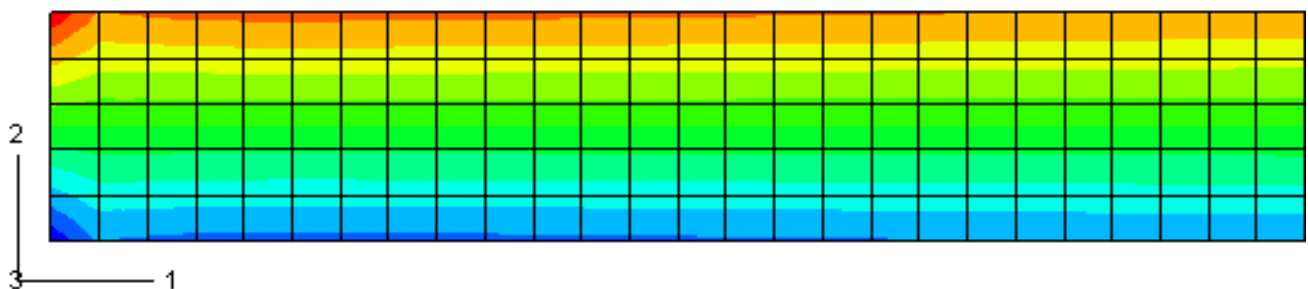
3. To plot the distribution of stress through the beam at *x = 0.1*, complete the following set of tasks:

4. From the main menu bar select **Tools→Path→Create**. The **Create Path** dialogue box will open as in **Fig.A3**.

    (a) Name the path `Stress Path`.

    (b) Choose **Node List** for the **Type**.

    (c) Click **Continue**.

    (d) From the **Edit Path List** dialogue box that opens select **Add Before**.
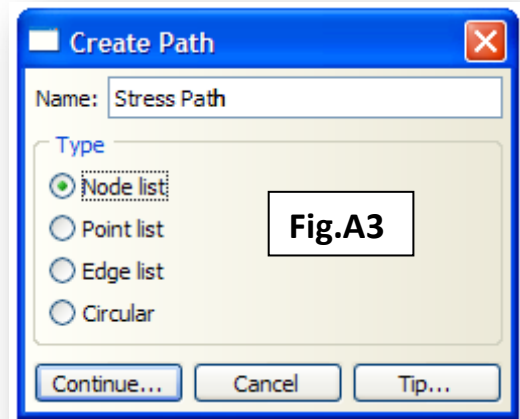
5. The length of the beam is 1 m. Since the beam has been meshed with 200 elements along the edge, the position x = 0.1, is equal to 20 elements along the beam. Using the cursor, select the nodes across the beam at x = 0.1 as shown. Click **OK** in the **Edit Path List** dialogue box.
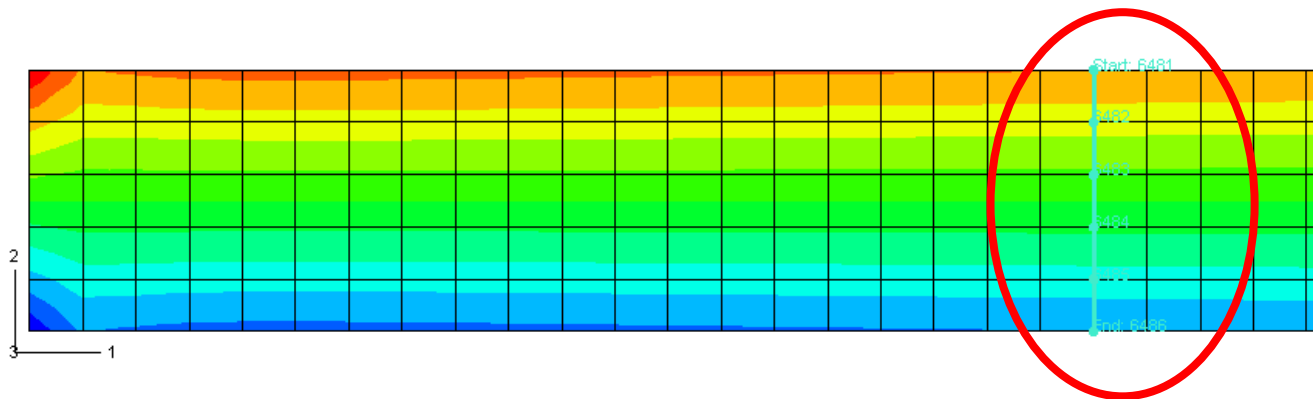
6. From the main menu bar select **Tools→XY Data→Create**. The **Create XY Data** dialogue box will open.

    (a) Select **Path** and click **Continue**.

    (b) Choose `Stress Path` as the path from the **XY Data from Path** dialogue box.

    (c) Choose **Deformed** for the **Model Shape**.

    (d) Under **X Values**. Toggle on **Y Distance**.

    (e) Click the **Field Output** icon and the **Field Output** dialogue box will open.

    (f) Select **S** as the primary variable and **S11** from the **Component** options.
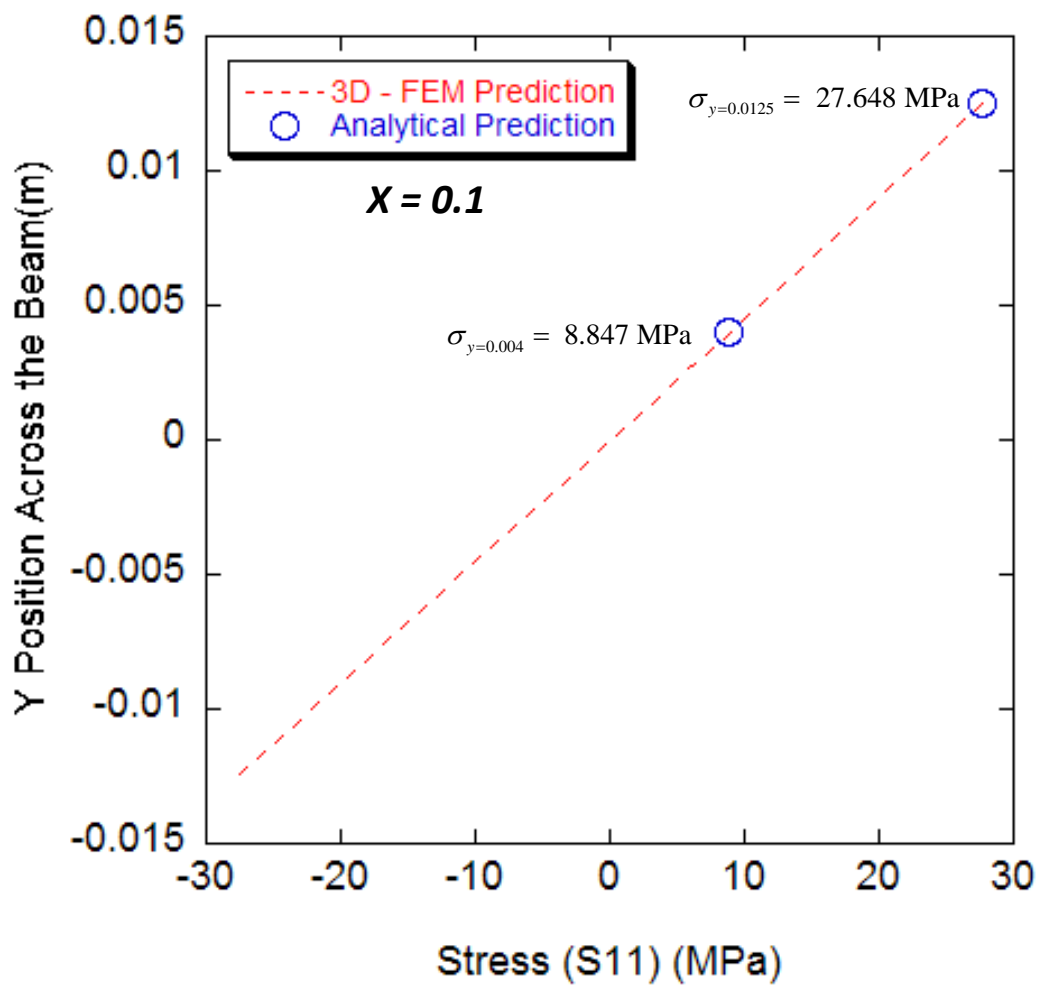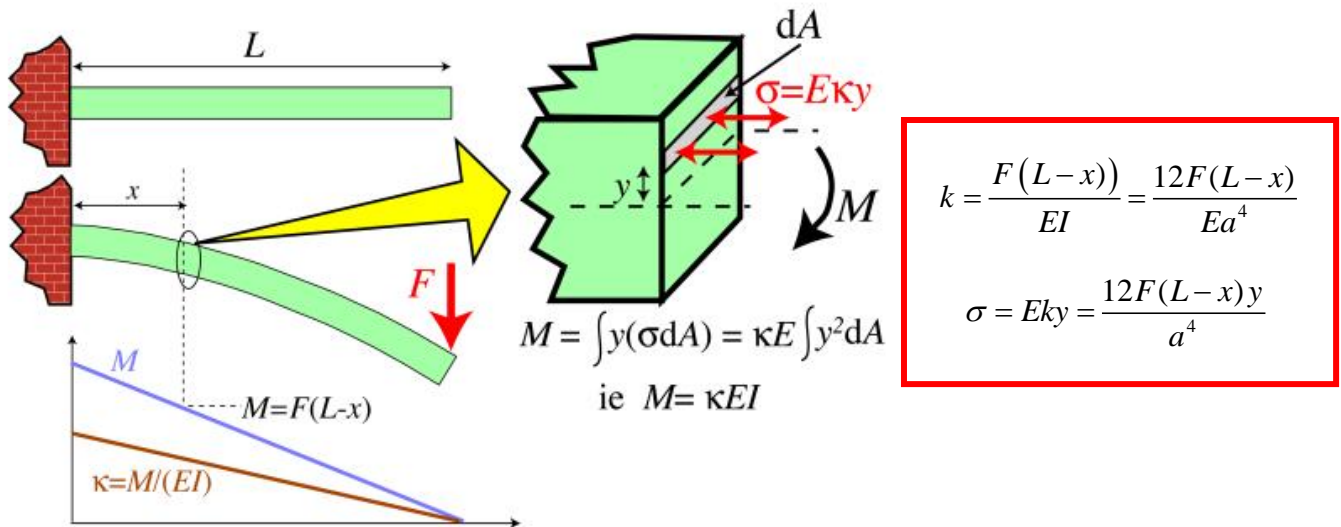
(g) Click Plot.

(h) Save the data. To access the data, from the main menu bar select **Tools→XY Data→Edit**.
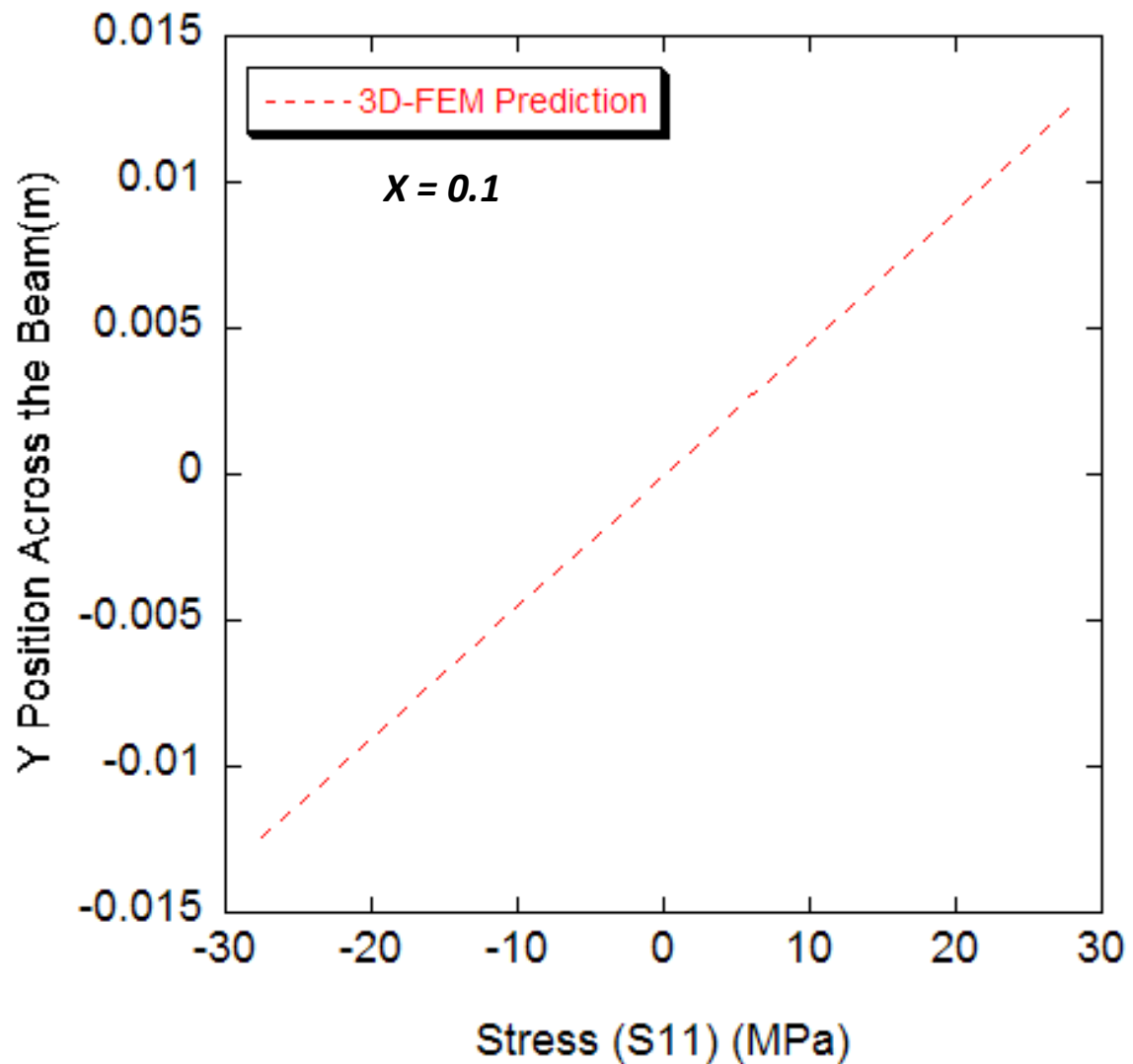
Is there an easier way to obtain these data? If you have time, think about creating a node set at x = 0.1.

Solution (f)

- Compare these stress data with those from ordinary beam bending theory at *x = 0.1* for *y = 0.004* and *y = 0.0125.*

$$k = \frac{F(L-x))}{EI} = \frac{12F(L-x)}{Ea^4}$$

$$\sigma = Eky = \frac{12F(L-x)y}{a^4}$$

OPTIONAL TASKS

---

1. Assume now that the material behaves in an elastic – perfectly plastic fashion, and that the yield stress is 150 MPa. Using the 3-dimensional finite element model, compute the deflection of the cantilever beam when loaded at its end with a force of 600 N. (*Hint; you will have to define this plastic material behaviour in your material model.)

2. Using the 3-dimensional finite element model, and assuming elastic - perfectly plastic behaviour, plot the **residual curvature** of the cantilever beam after it has been loaded at its end with a force of 600 N. Compare the deflected profile from Question 1 with this residual curvature. (*Hint; Think about creating a second step in which the load from the `Load Step` is not **Propagated**!)

3. Assume now that the aluminium exhibits linear work hardening behaviour, with a work hardening rate $d\sigma/d\varepsilon$ of 300 MPa. Compute the deflection of the beam when loaded at its end with a force of 600 N. Compute also the deflection of the beam when the load is removed. Compare the results with those of the perfectly plastic case.

(*Hint; σ_Y = 150 MPa at $\varepsilon_{plastic}$ = 0 and σ_Y = 450 MPa at $\varepsilon_{plastic}$ = 1)

4. Assume now that the aluminium hardens according to a power-law relationship of the following kind: $\sigma = \sigma_y + k\varepsilon_{plastic}^{n}$, where the yield stress is 150 MPa, k is a constant equal to 300 MPa and the hardening exponent, *n*, is 0.4. Calculate σ for $0 < \varepsilon_{plastic} < 1$. Using this data, compute the deflection of the cantilever beam, loaded at its end with a force of 600 N. Find also the residual curvature of the beam. Compare these data with the linear work-hardening case.

Heat Transfer Analysis

Type of solver: ABAQUS CAE/Standard

(A) Two-Dimensional Steady-State Problem – Heat Transfer through Two Walls

Problem Description:

The figure below depicts the cross-sectional view of a furnace constructed from two materials. The inner wall is made of concrete with a thermal conductivity of $k_c = 0.01$ W m^{-1} K^{-1}. The outer wall is made of bricks with a thermal conductivity of $k_b = 0.0057$ W m^{-1} K^{-1}. The temperature in the furnace is at 1273 K and the convective heat transfer coefficient is $h_1 = 0.208$ W m^{-2} K^{-1}. The outer brick wall comes into contact with the ambient air, which is at 293 K, and the corresponding convective heat transfer coefficient is $h_2 = 0.068$ W m^{-2} K^{-1}.

Formulate a 2-D FE model and solve for (i) the temperature distribution within the concrete and brick walls at steady-state conditions, and (ii) the heat flux across the walls.
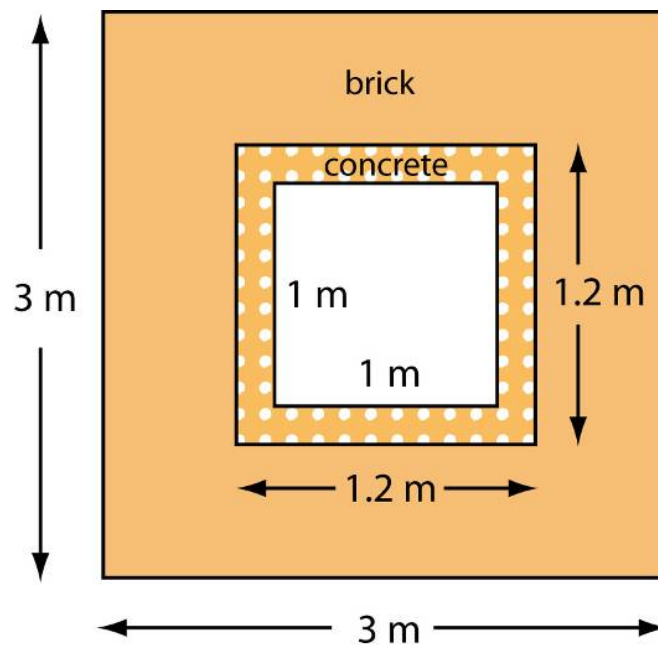
SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.
- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model 2D_Walls

A. MODULE → PART

Under the Part module, we will construct the two parts (i.e. walls): (i) Brick and (ii) Concrete

1. From the main menu bar, select **Part→Create**

2. The **Create Part** dialog box appears. Name the part Brick and fill in the rest of the options as in **Fig.A1**. Click **Continue** to create the part.

3. There are several ways of constructing the brick wall geometry.

One way to do this is demonstrated here:

(a) From the Sketcher toolbox, select the **Create Isolated Point** tool ⊞, then type in coordinates of the four key vertices (0, 0), (0.9, 0.9), (2.1, 2.1) and (3, 3). If not all plotted points are visible, press the **Auto Fit View** button located on the toolbar.

(b) From the Sketcher toolbox, select the **Create Lines: Rectangle** tool and connect the inner and outer pairs of vertices to form two squares, as shown in **Fig.A2**.

(c) Click on **Done** in the prompt area.

Fig.A1

Fig.A2

44
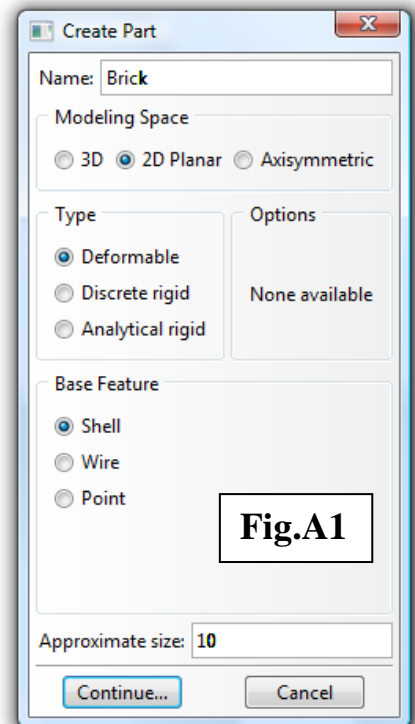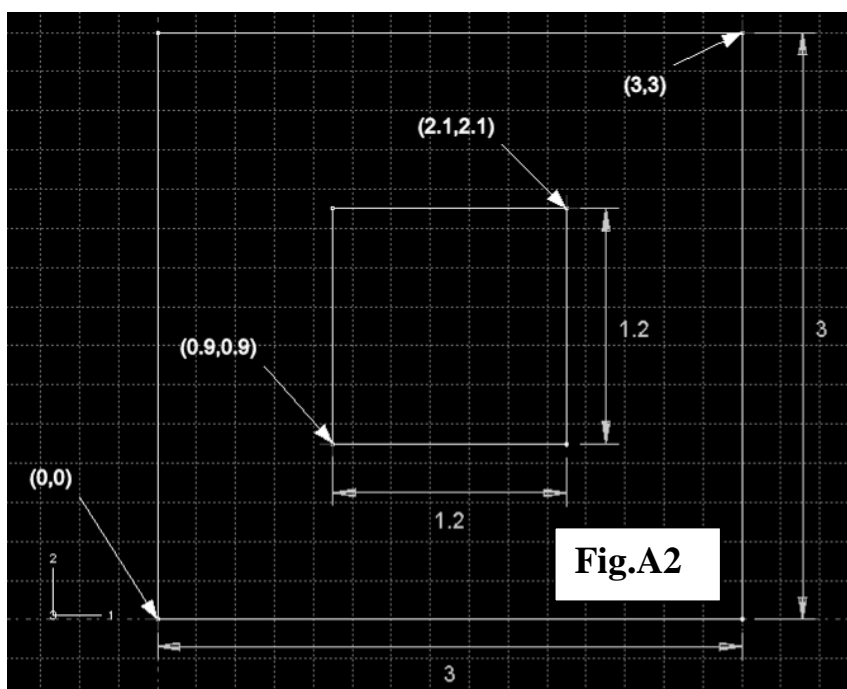
4. Now construct the second part by following procedures similar to the ones outlined above. Name the new part `Concrete`. The four key vertices are (0, 0), (0.1, 0.1), (1.1, 1.1) and (1.2, 1.2).

B. MODULE → PROPERTY

(a) To define the materials:-

1. From the main menu bar, select **Material→Create**

2. The **Edit Material** dialog box appears (see **Fig.B1**). Name it `Material-brick`. Select **Thermal→Conductivity** and enter a value of `0.0057`.

3. Click **OK**.

4. Now create `Material-concrete`. Enter a value of 0.01 as its thermal conductivity.
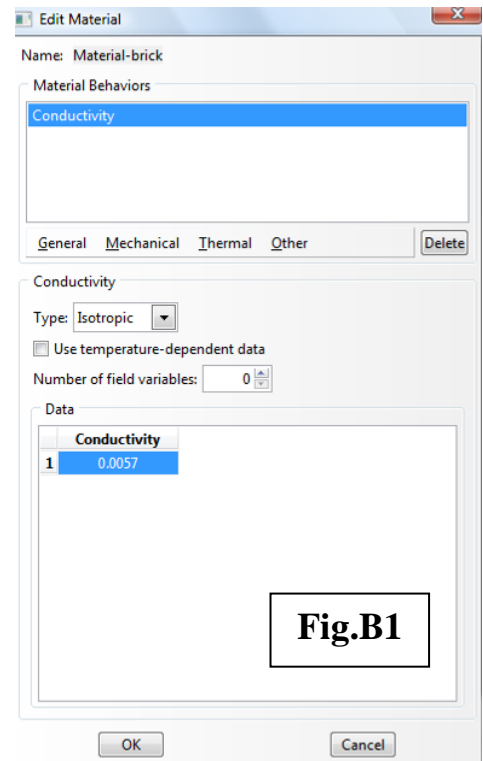
(b) To define the sections:-

1. From the main menu bar, select **Section→Create**

2. The **Create Section** dialog box appears (**Fig.B2**). Name it `Section-brick`. In the **Category** list, accept **Solid** as the default selection. In the **Type** list, accept **Homogeneous** as the default selection, and click **Continue**.

3. The **section editor** appears (**Fig.B3**). Click the arrow next to the **Material** text box and choose `Material-brick`. Accept the default value for **Plane stress/strain thickness**, and click **OK.**

4. Now define `Section-concrete.`
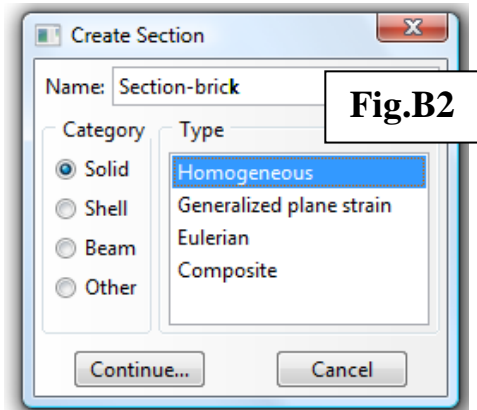
(c) To assign a section to a part:-

1. From the main menu bar, select **Assign→Section**

2. Click on the Brick region and then click **Done**.

3. The **Edit Section Assignment** dialog box appears containing a list of existing sections, Click the arrow next to the **Section** text box and choose `Section-brick`,

Fig.B1

Fig.B2
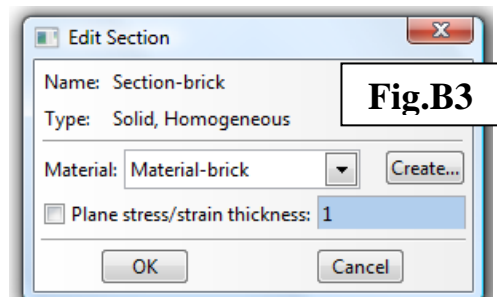
Fig.B3

and click **OK**.

Note: the colour of a part becomes *aqua* when it has been assigned a section.

5. Now assign `Section-concrete` to the concrete region.

C. MODULE → ASSEMBLY

1. From the main menu bar, select **Instance→Create**

2. The **Create Instance** dialog box appears (**Fig.C1**). Under **Parts**, select `Brick`. For **Instance Type**, choose **Independent (mesh on instance)**. Toggle on **Auto-offset from other instances**. Click **OK**.
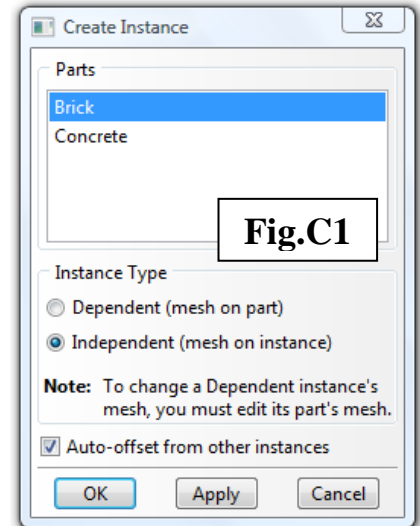
3. Now create an instance for the part `Concrete`.

4. At this point, before we proceed onto assembling the instances, it would be useful to define several sets of surfaces for use in later stages of the analysis. From the main menu bar, select **Tools→Surface→Create**. The **Create Surface** dialog box appears. Name it `Brick-inside` and pick the four edges located inside the `Brick` instance, see **Fig.C2** (*Note:* you may need to press and hold the *Shift*-key to make multiple selections). Click **Done** in the prompt area. Repeat to create another set of surface called `Brick-outside`, consisting of four edges located outside the Brick instance, see **Fig.C2**.

5. Now create the following surfaces on the Concrete instance, name them: `Concrete-inside` and `Concrete-outside`, corresponding to the four inner and outer edges of the Concrete instance, as depicted in **Fig.C2**.
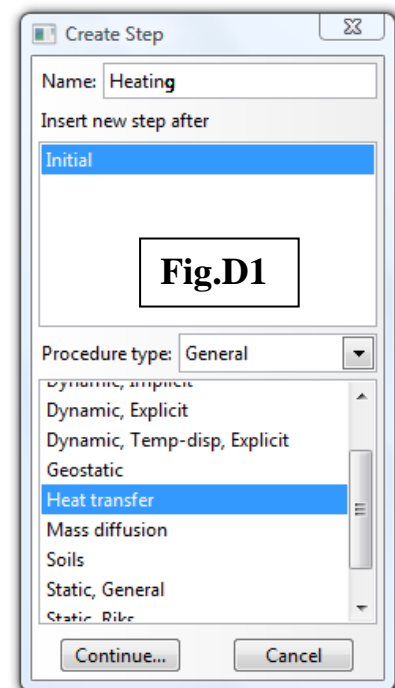

Fig.C1

Fig.C2
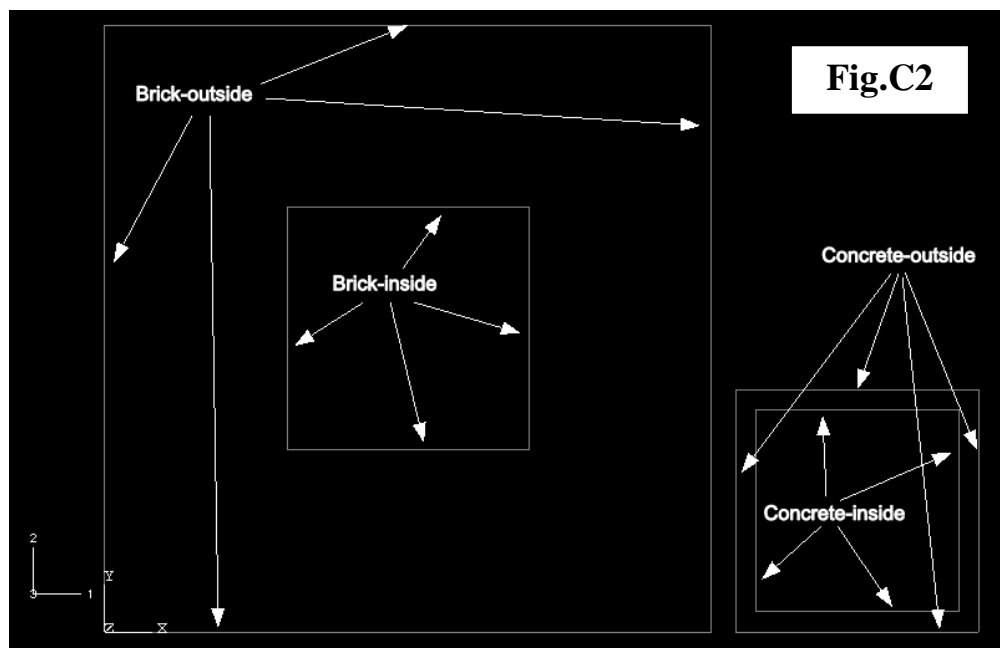
6. We'll now assemble the two instances. From the main menu bar, select **Instance→Translate.** Select the `Concrete` instance and click **Done**. By picking the suitable start and end points for the translation vector, position the smaller concrete wall within the larger brick wall, so that the final assembly resembles **Fig.C3**.

Fig.C3
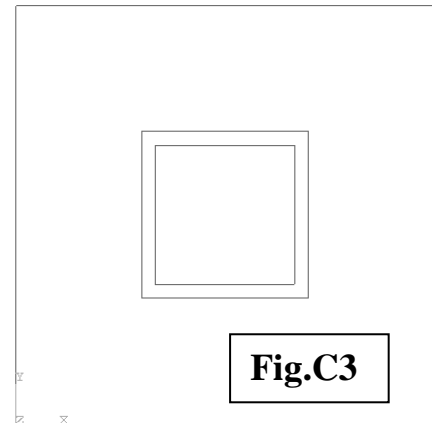
D. MODULE → STEP

1. From the main menu bar, select **Step→Create**

2. The **Create Step** dialog box appears (**Fig.D1**), name it `Heating`, and select **Heat transfer** under **Procedure type**. Click **Continue**.

3. The **Edit Step** dialog box appears. Under the **Basic** tab, toggle on **Steady-state**, click **OK**.

4. From the main menu bar, select **Output→History Output Requests→Create**, accept the default name `H-Output-1`, the **Edit History Output** dialogue box appears, expand the **Thermal** button and toggle on **FTEMP**. Click **OK**.

Fig.D1

Create Step

Name: Heating

Insert new step after

Initial

Procedure type: General

Dynamic, Implicit
Dynamic, Explicit
Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils
Static, General
Static, Riks

Continue...   Cancel

E. MODULE → INTERACTION

(a)To tie the nodes at the interfaces:-

1. From the main menu bar, select **Constraint→Create**

2. The **Create Constraint** dialog box appears, name it `Interface` and under **Type** pick **Tie**. Click **Continue**.

   Note: since we assume there is no thermal resistance across the brick-concrete wall interface, the **Tie** constraint will equate temperatures at the matching nodes.

3. In the prompt area, choose the master type as **Surface**, click on the `Surfaces...` button at the lower right hand corner of the prompt area. Select `Brick-inside` and click **Continue**. Click the **Surface** button in the prompt area and select `Concrete-outside` as the slave surface. Click **Continue**.

4. The **Edit Constraint** dialog box appears (**Fig.E1**), accept the default settings and click **OK**.

(b) To assign convective heat transfer conditions:-

1. From the main menu bar, select **Interaction→Create**

2. The **Create Interaction** dialog box appears (**Fig.E2**), name it `Int-InnerWalls`. Under **Step**, choose `Heating`. For **Types for Selected Step**, choose **Surface film condition**, click **Continue**. In the **Region Selection** dialog box, select the surface defined earlier as `Concrete-inside` and click **Continue**.

    Note: if the **Region Selection** dialog box does not appear, click on the `Surfaces...` button at the bottom right hand corner of the prompt area.

3. The **Edit Interaction** dialog box appears (**Fig.E3**), enter 0.208 (W m^{-2} K^{-1}) as the **Film coefficient**, and 1273 (K) as the **Sink temperature**.

4. Now create surface film condition for the brick walls that are in contact with the ambient air, name it `Int-OuterWalls`. Apply it to the surface called `Brick-outside.` Enter 0.068 (W m^{-2} K^{-1}) as the **Film coefficient**, and 293 (K) as the **Sink temperature**.

48
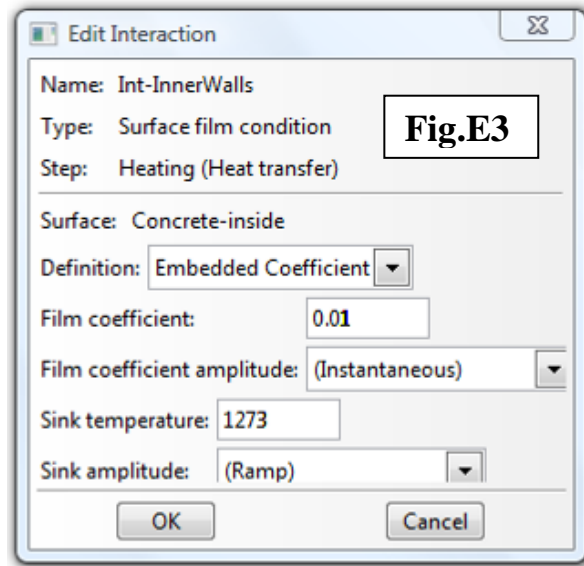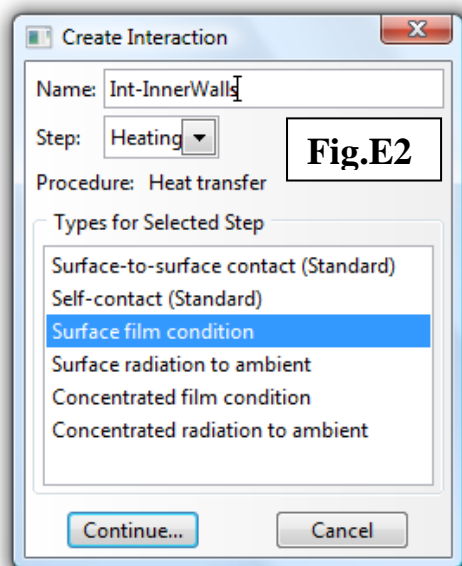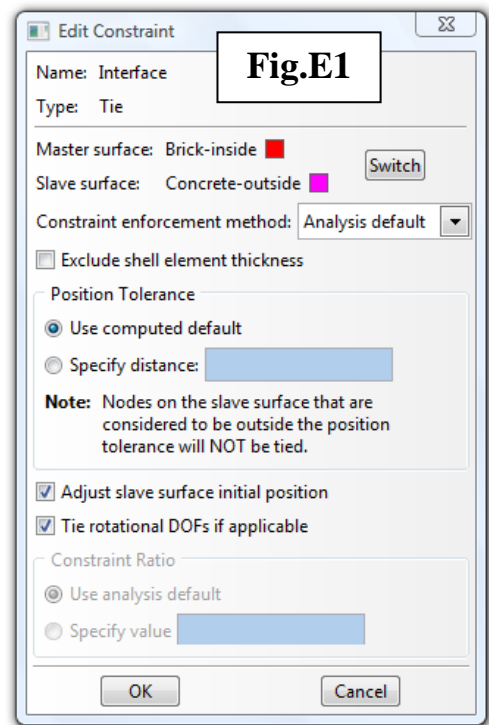
F. MODULE → MESH

*(a) To seed the part instance:-*

1. From the main menu bar, select **Seed→Instance**

2. Left click on the `Brick` region, click **Done** in prompt area. The Global Seeds dialog box appears, enter `0.1` for **Approximate global size**, accept the rest of the settings and click **OK**.

3. By following the above steps, now apply an **Approximate global seed size** of `0.02` to the `Concrete` region.

(b) To assign mesh controls:-

1. From the main menu bar, select **Mesh→Controls**

2. Select both the `Brick` and `Concrete` regions, you can do this by dragging a box across them. Click **Done** (on the prompt area).

Fig.F1

3. The **Mesh Controls** dialog box appears, follow the settings depicted in **Fig.F1**. Ensure that **Medial axis** algorithm is chosen.
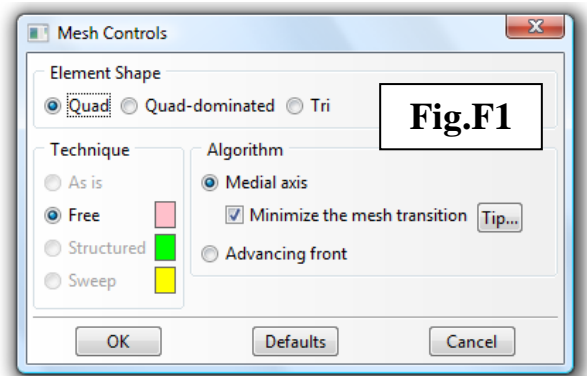
(c) To assign element type:-

1. From the main menu bar, select **Mesh→Element Type**

2. Select both regions. Click **Done**.

3. The **Element Type** dialog box appears (**Fig.F2**), under the **Family** list, ensure that **Heat transfer** is selected. The element type to be assigned is **DC2D4**.
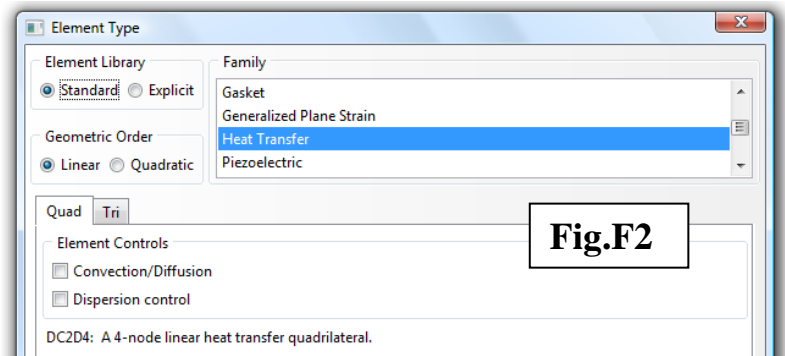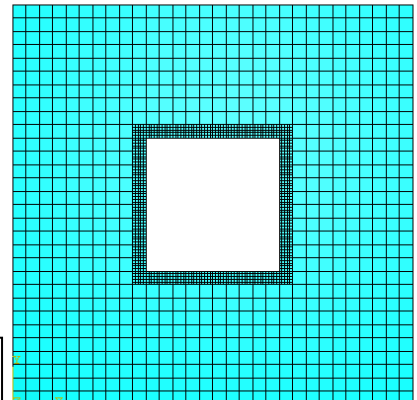

Fig.F2

(d) To mesh the part instance:-

1. From the main menu bar, select **Mesh→Instance**

2. Select both regions. Click **Done**. The generated mesh should resemble **Fig.F3**.

Fig.F3

G. MODULE → JOB

(a) To create a new job:-

1. From the main menu bar, select **Job→Create**

2. The **Create Job** dialog box appears, enter `Job-2D-Thermal`. Click **Continue**.

3. The **Edit Job** dialog box appears, accept the default settings and click **OK**.

(b) To submit the job:-

1. From the main menu bar, select **Job→Manager**

2. The **Job Manager** dialog box appears (**Fig.G1**), select `Job-2D-Thermal` and click on the **Submit** button. To see the progress of the analysis, and to monitor error and warning messages, click the **Monitor** button to bring up the **Monitor** dialog box (**Fig.G2**).
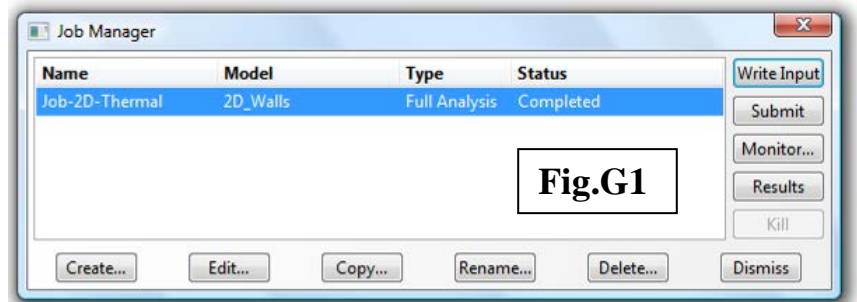
Fig.G1

(c) To analyse the results:-

When the job is **Completed**, click on the **Results** button on the **Job Manager** dialog box (**Fig.G1**).
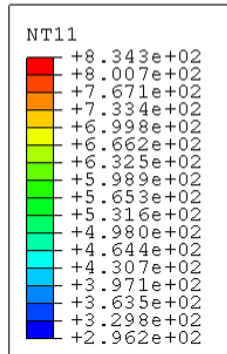
Note: If the job fails to complete, go back to the **Monitor** dialog box (**Fig.G2**) and examine the messages under **Errors** and **Warnings** tabs, which often will provide clues on how to troubleshoot the analysis.
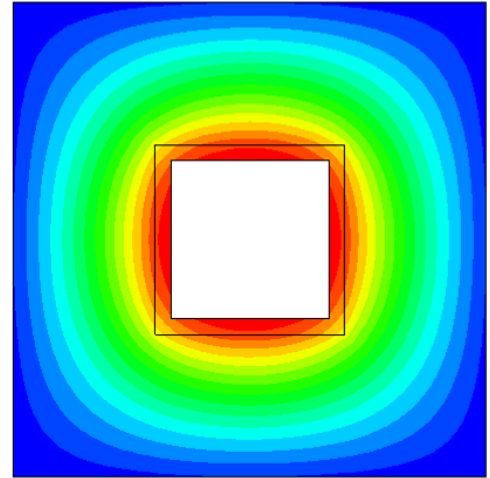

Fig.G2
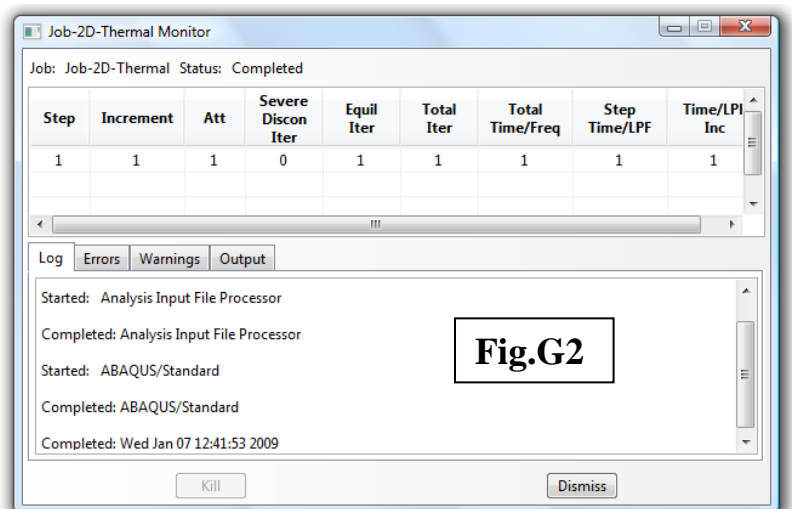
H. MODULE → VISUALIZATION

1. From the main menu bar, select **Results→Field Output**

2. The **Field Output** dialog box appears, under **Primary Variable**, select **NT11** and click **OK** to produce the nodal temperature distribution plot (**Fig.H1**). *Note:* The temperature is in Kelvin.


Fig.H1

3. Click on **Contour Options** on the prompt area, and you'll be presented with various ways of customising the output.
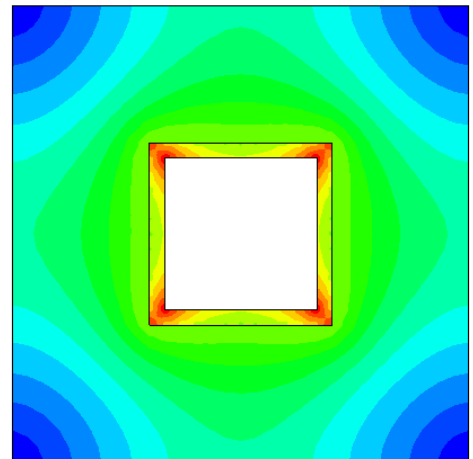
4. To display the heat flux distribution within the walls, from the main menu bar, select **Results→Field Output**

5. The **Field Output** dialog box appears, under **Primary Variable**, select **HFL** and click **OK.** The heat fluxes shown here are in W m^{-2} (**Fig.H2**), consistent with the SI unit we employed while setting up the model.
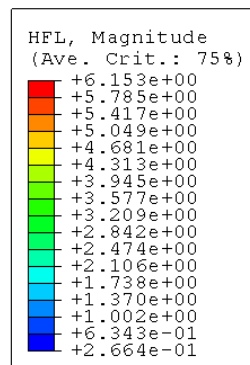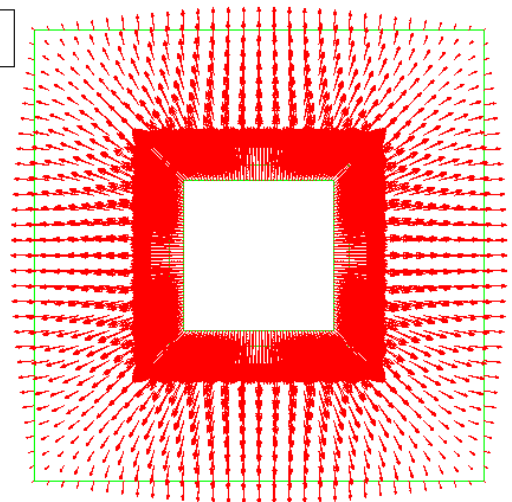

Fig.H2

6. To generate a resultant vector plot of the heat fluxes (**Fig.H3**), from the main menu bar, select **Plot→Symbols**. Click on **Symbol Options** on the prompt area to customise the vector plot.
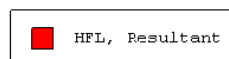
Fig.H3

TASKS

1. Due to symmetry, it is possible to model a quarter or just one-eighth of the system by applying suitable boundary conditions. Demonstrate how this could be done in ABAQUS.

2. When it comes to meshing, there are many possibilities in terms of the choice of mesh size and density, element type, shape and order, and meshing technique or algorithm. Explore how some of the above can affect the accuracy of your model predictions.

3. Compare the variation of thermal gradients across the different sections of the walls (e.g. along the horizontal, vertical, diagonal directions etc.).

(B) Three-Dimensional Transient Problem – Heat Dissipation through Ribbed Surfaces

Problem Description:

Ribbed surfaces or fins are commonly used in engineering applications to dissipate heat. The figure below shows the 2-D cross-section and 3-D geometry of an aluminium heat sink designed for cooling a microprocessor. The thermal conductivity of aluminium is $k = 170$ W m^{-1} K^{-1}. The initial temperature of the heat sink is 293 K. When the microprocessor is operating, the bottom surface of the heat sink is exposed to a constant heat flux of $q = 1000$ W m^{-2}. Forced air flow from a cooling fan over the developed surface maintains the surrounding surface at 323 K. The convective heat transfer coefficient between the fin and the ambient surrounding is at $h = 80$ W m^{-2} K^{-1}.

Formulate a transient 3-D FE model to predict (i) the time needed for the heat sink to achieve steady-state conditions, and (ii) the temperature distribution within the developed surfaces.

SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.

- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model 3D_Fin

---

A. MODULE → PART

1. From the main menu bar, select **Part→Create**

2. Name the part Fin and follow the settings depicted in **Fig.A1**. The approximate size is set at 0.1 (metre).

3. Sketch the 2-D profile (**Fig.A2**) according to the dimensions given in the *Problem Description*.

   Note: Remember to construct the model in SI units.

   Tips: (a) To ease sketching, click on the **Sketcher Options** tool located in the Sketcher toolbox and change the **Grid spacing** to 0.001 and the **Minor Intervals** to 1.
   (b) You can exploit the symmetry by using the "Mirror" tool, located under **Edit→Transform→Mirror**.

4. When done sketching, click **Done** in the prompt area. The **Edit Base Extrusion** dialog box appears, enter the base extrusion depth as 0.02

Fig.A1

Fig.A2
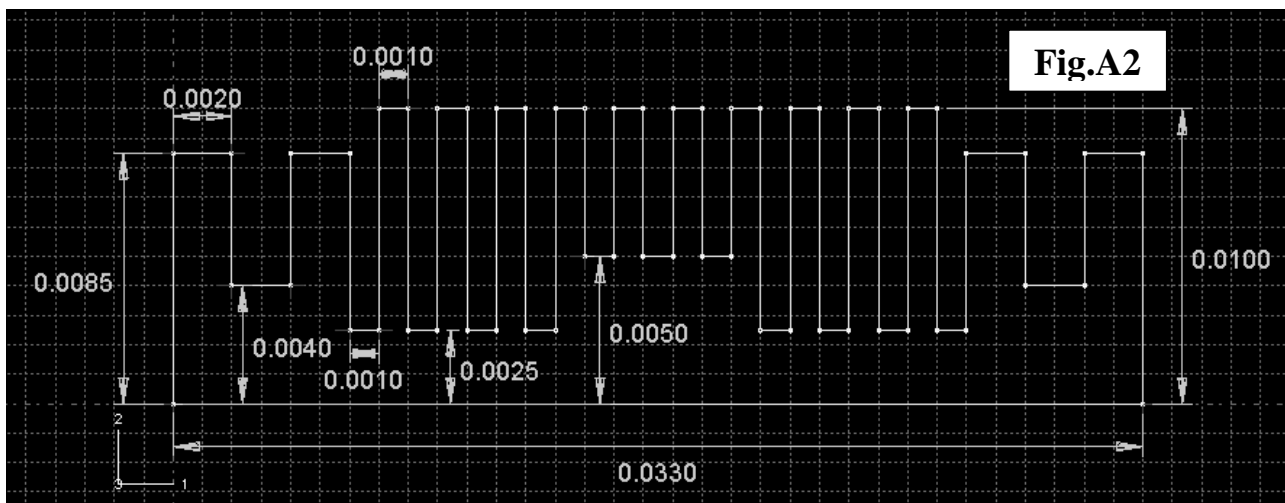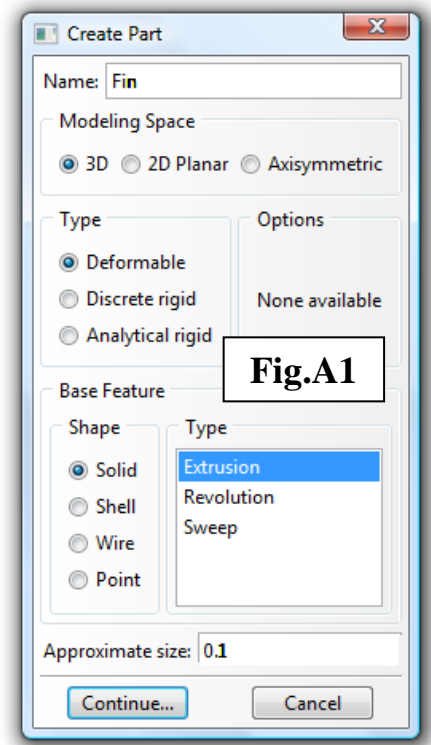
B. MODULE → PROPERTY

1. From the main menu bar, select **Material→Create**

2. Name the material `Aluminium`.

3. Create the following material properties (**Fig.B1**):-
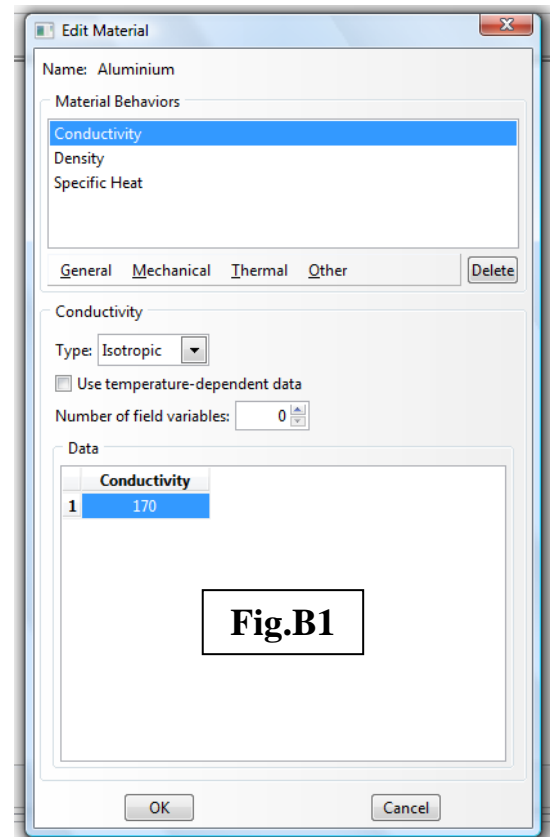
   General→Density 2700 kg m^{-3}

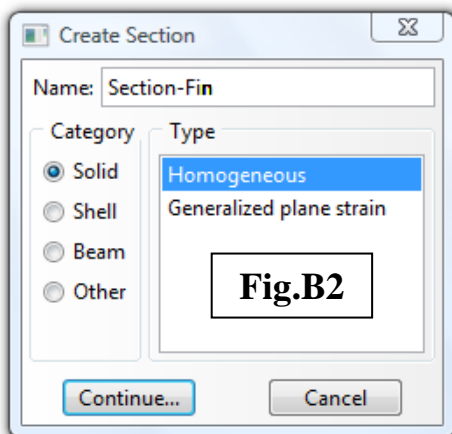   **Thermal→Conductivity** $170 \text{ W m}^{-1} \text{ K}^{-1}$

   Thermal→Specific Heat $950 \text{ J kg}^{-1} \text{ K}^{-1}$

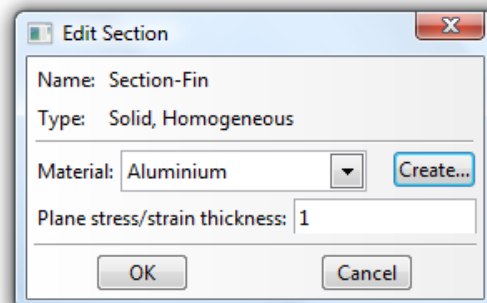   *Note:* Since this will be transient heat transfer analysis, we need to include both density and specific heat properties.

4. Create a new section, name it `Section-Fin`, use the settings as shown in **Fig.B2** and **Fig.B3.**

5. Assign the section to the `Fin` part.


Fig.B1

Fig.B2

Fig.B3

C. MODULE → ASSEMBLY

1. From the main menu bar, select **Instance→Create**

2. Create an instance of the `Fin` part. Under **Instance Type**, make sure to select **Independent (mesh on instance)**.

D. MODULE → STEP

*(a) To create the transient analysis step*

1. From the main menu bar, select **Step→Create**

2. Name it `Transient-heating`. The **Procedure type** is **General→Heat Transfer**

3. In the **Edit step** dialog box
   (**Fig.D1**), under the **Basic** tab,
   ensure that the **Response** is
   Transient, and set the **Time period**
   as `5000` (seconds).
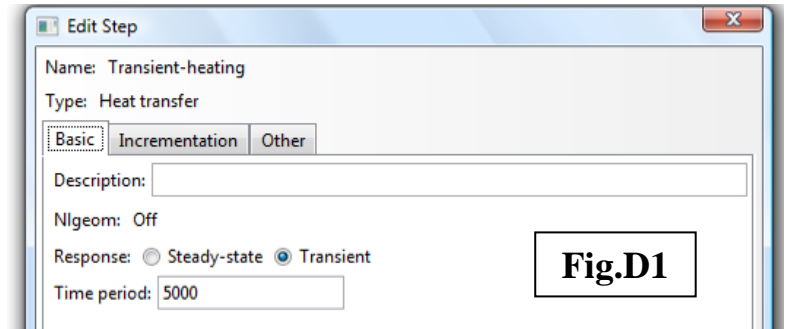
4. Click on the **Incrementation** tab
   (**Fig.D2**), increase the **Maximum number of increments** to `200`. Change the **Initial**
   Increment size to `0.1` and
   Maximum Increment size to `100`.

5. Toggle on **End step when**
   temperature change is less than
   and enter `0.0001`, so that iteration
   will stop once thermal equilibrium
   is reached.

6. Set the **Max. allowable**
   temperature change per
   increment to 5 (Kelvin).

7. Accept the default settings under the **Other** tab.
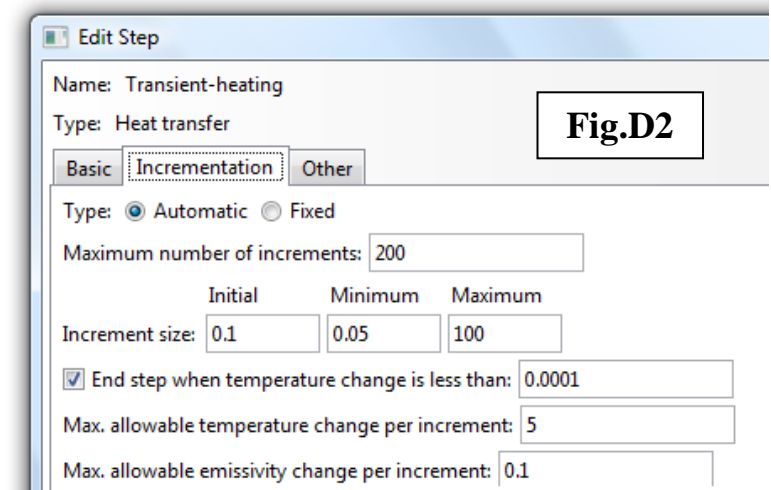
(b) To edit the field output

1. From the main menu bar, select **Output→Field Output Requests→Edit→**`F-Output-1`

2. Under **Output Variables**, toggle on **Thermal** and select **NT** and **HFL**.

(c) To edit the history output

1. First we create a node set to record the temperature history. From the main menu bar, select **Tools→Set→Create**. The **Create Set** dialog box appears, name it `Set-Node1` and pick the node depicted in **Fig.D3**.

Set-Node1

Fig.D3

2. From the main menu bar, select **Output→History Output Request→Create→**`H-Output-1`

3. The **Edit History Output Request** dialogue box appears (**Fig.D4**). Change the **Domain** to **Set** and choose `Set-Node1`. Save output at **Every 1** increments. Under **Output Variables**, choose **Thermal→NT, Nodal temperature**

Fig.D4
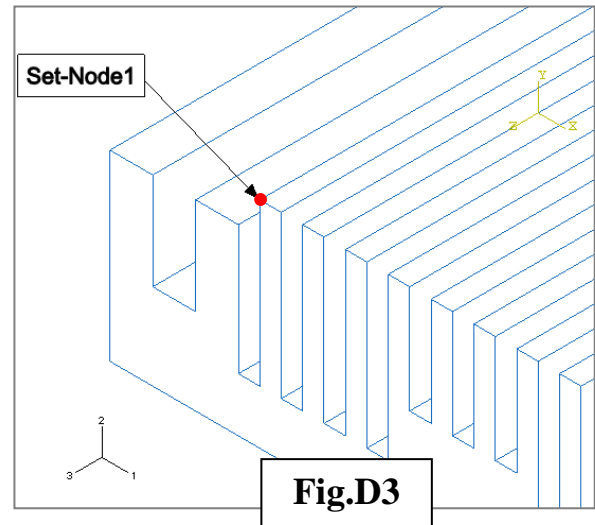
(d) Create a DOF monitor

1. A Degree of Freedom (DOF) monitor is useful to follow the progress of a transient analysis. Here, we'll set up `Set-Node1` to monitor the temperature evolution.

2. From the main menu bar, select **Output→DOF Monitor**

3. Fill out the options as in **Fig.D5**. Note that DOF 11 corresponds to temperature in ABAQUS/CAE.

Fig.D5

57
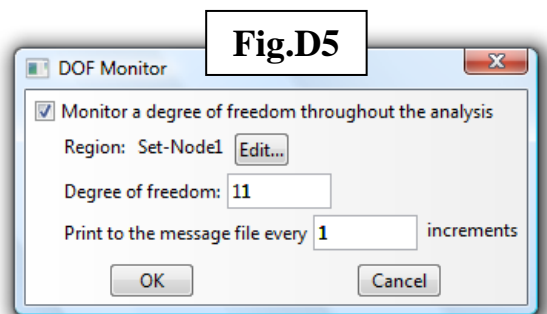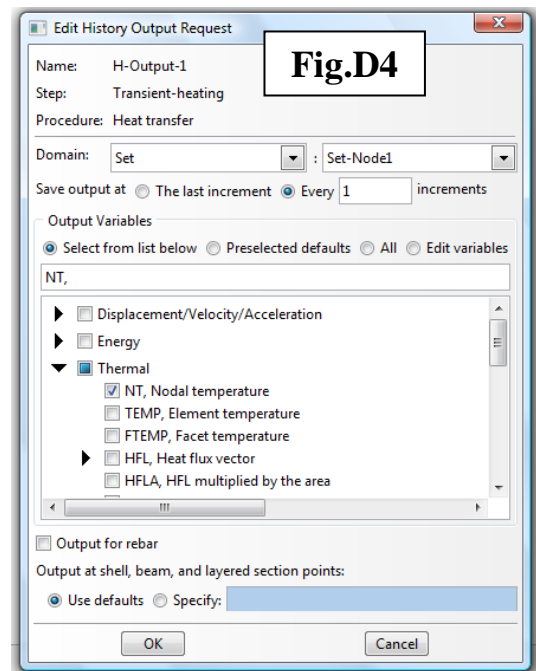
E. MODULE → INTERACTION

1. From the main menu bar, select **Interaction→Create**

2. Name it `Int-Convection`. Under **Types for Selected Step**, choose **Surface film condition**, see **Fig.E1**.
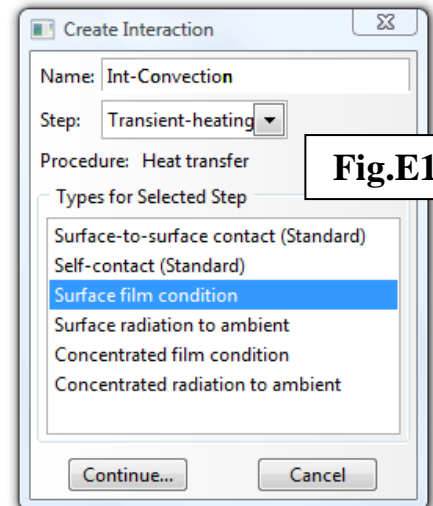

Fig.E1

3. The next task is to select the surfaces to apply the film conditions. However, since there are so many surfaces involved, it will be more convenient to do it as follows:-

   (a) From the main menu bar, select

   View→Toolbars→Views

   (b) Click the **Apply Front View** button:

   (c) Now drag a box across the screen to pick all surfaces above the base surface (as indicated by dotted lines in **Fig.E2**). *Important*: Ensure that all surfaces are selected except the base.
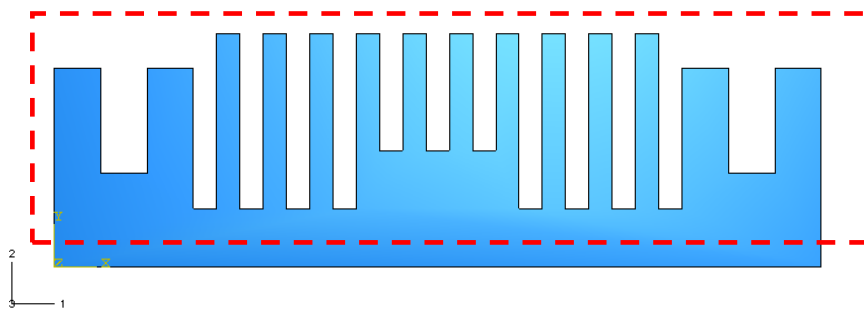

Fig.E2

   (d) The **Edit Interaction** dialog box appears (**Fig.E3**), fill in the **Film coefficient** as `80` (K) and set the **Sink temperature** as `323` (W m^{-2} K^{-1}).
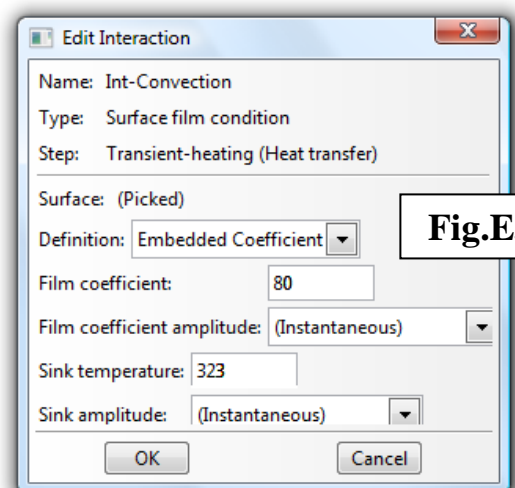

Fig.E3

F. MODULE → LOAD

*(a) To create load (i.e. heat flux at the base of heat sink)*

1. From the main menu bar, select **Load→Create**

2. Name it `Apply_Heatflux`. Under **Types for Selected Step**, choose **Surface heat flux**, see **Fig.F1**.

3. When prompted to choose the surface for the surface heat flux, it would be necessary to rotate the view ↺ so that the bottom surface of the heat sink can be selected (**Fig.F2**).


Fig.F2
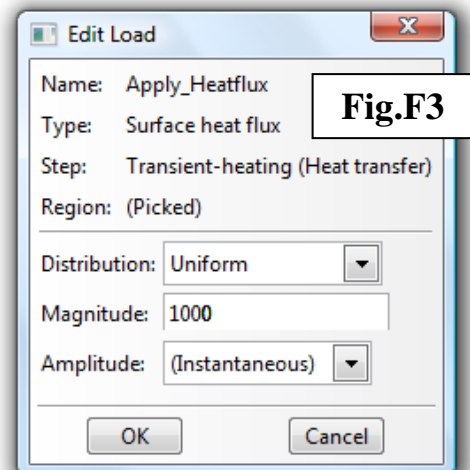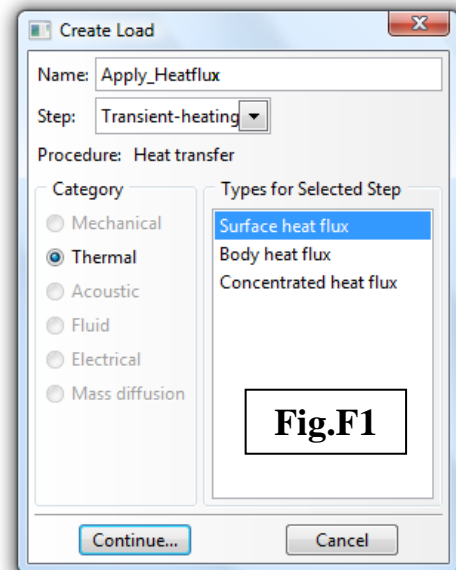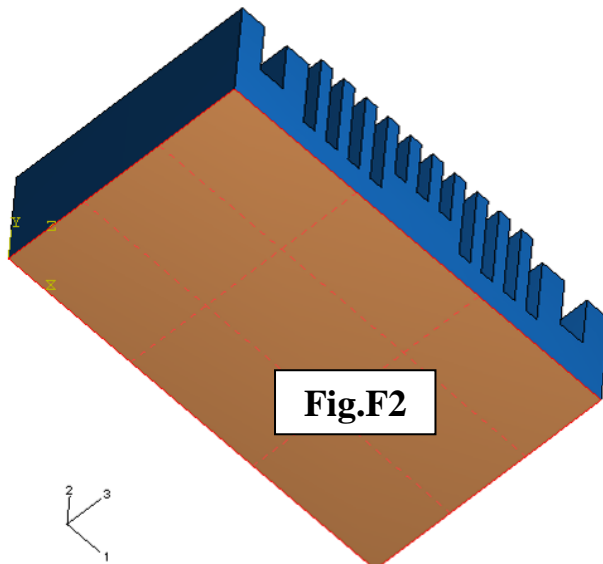
4. Fill out the **Edit Load** dialog box as in **Fig.F3**.

(b) To create field (i.e. initial temperature)

1. From the main menu bar, select **Predefined Field→Create**

2. Name it `Initial-Temperature`. Under **Step**, choose **Initial**. Under **Category**, choose **Other→Temperature**, see **Fig.F4**.

Fig.F1

Fig.F3

Fig.F4
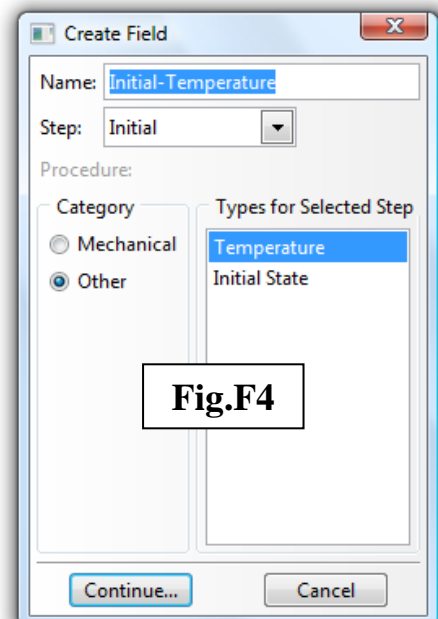
3. When prompted to select region for the field, drag a box across the whole assembly to select all surfaces. Fill out the **Edit Filed** dialogue box as in **Fig.F5**.
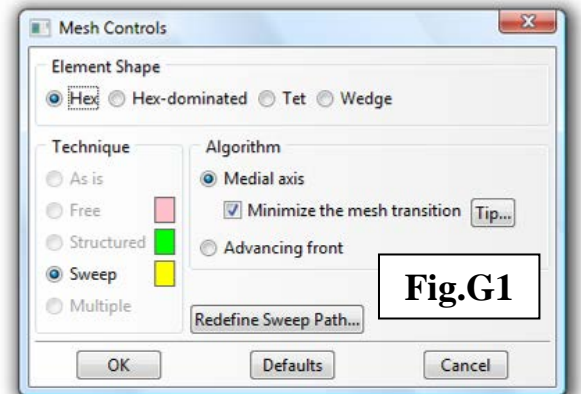

Fig.F5
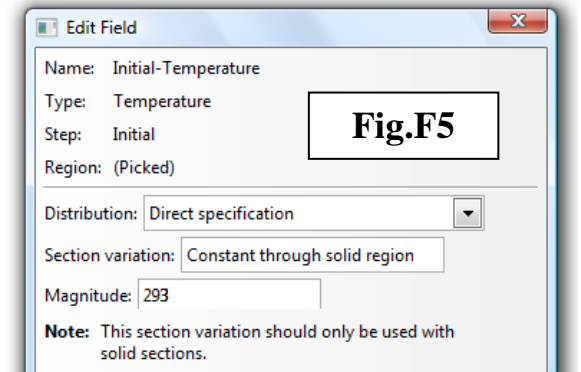
G. MODULE → MESH

(a) To seed the part instance:-

1. From the main menu bar, select **Seed→Instance**

2. Apply 0.0005 for the **Approximate global size**.
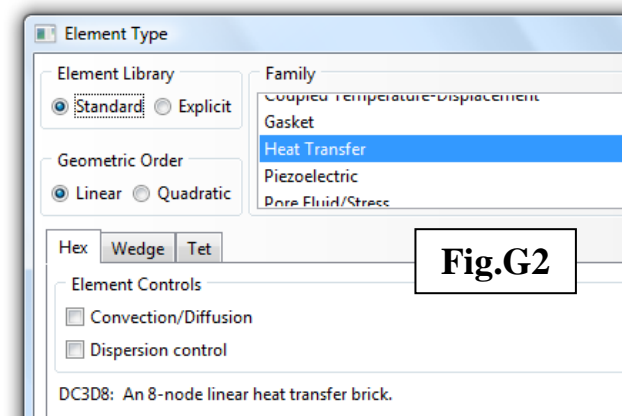
(b) To assign mesh controls:-

1. From the main menu bar, select **Mesh→Controls**

2. The **Mesh Controls** dialog box appears, follow the settings depicted in **Fig.G1**.

Fig.G1

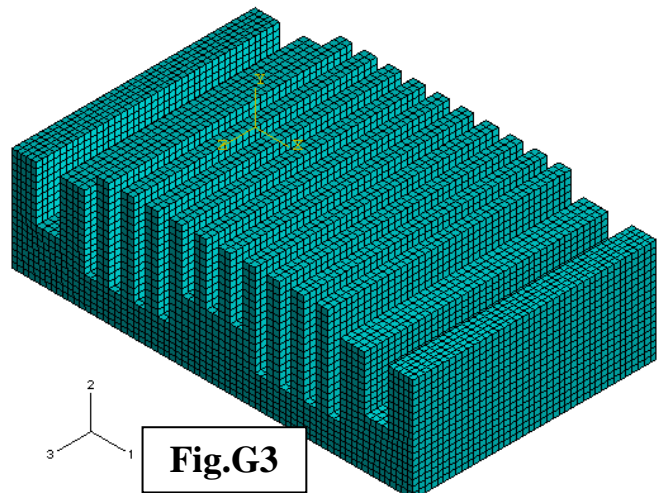*(c) To assign element type:-*

1. From the main menu bar, select **Mesh→Element Type**

2. The **Element Type** dialog box appears (**Fig.G2**), under the **Family** list, choose **Heat transfer**. The type of element assigned is **DC3D8**.

Fig.G2

(d) To mesh the part instance:-

1. From the main menu bar, select **Mesh→Instance**

2. The generated mesh should resemble **Fig.G3**.

Fig.G3

H. MODULE → JOB

1. From the main menu bar, select **Job→Create**

2. Enter `Job-3D-Fin` as the job name, ensure that the **Source Model** chosen is `3D_Fin.`

3. Submit the job and monitor the progress. Since this is a transient analysis, longer computation time is expected (may take 10 to 20 minutes depending on your system).

4. When the job is completed, from the **Job Manager** dialogue box, click on **Results**.

I. MODULE → VISUALIZATION

1. From the main menu bar, select **Results→History Output**. Plot the nodal temperature of `Set-Node1` as a function of time (**Fig.I1**). It can be seen that it takes about 500 s to reach steady-state conditions.

Fig.I1
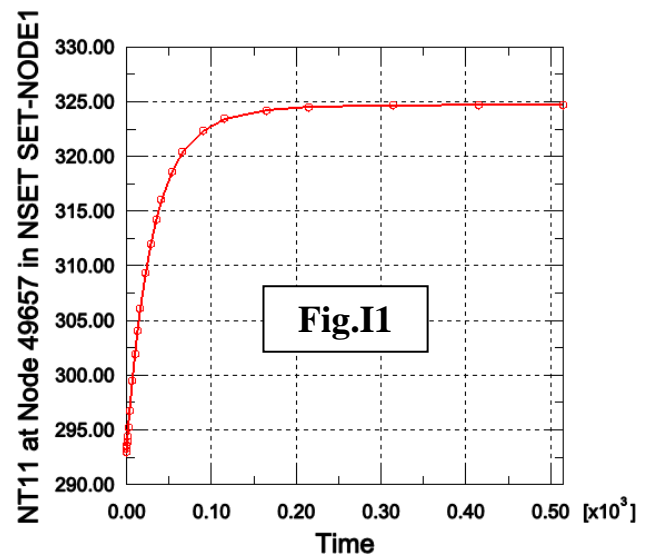
2. To display the nodal temperature distribution, from the main menu bar, select **Results→Field Output** and select **NT11. Fig.I2** shows the temperature field at steady-state.

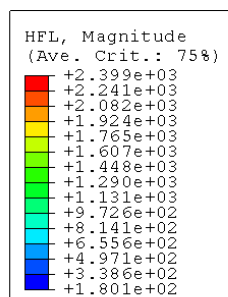*Note:* Using the control buttons ⏮ ◀ ▶ ⏭ 🔁 in the context bar, you can step though the frames to examine the temporal evolution of the thermal field.
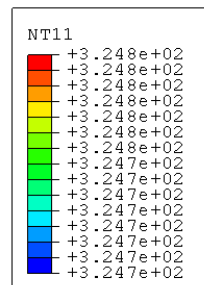
Fig.I2

3. To display the heat flux distribution, from the **Field Output** select **HFL**. **Fig.I3** shows the temperature field at the steady-state condition.

Fig.I3

J. TASKS

1. Instead of using a transient model, solve the above problem using a steady-state model.

2. Compute the temperature gradients across different sections of the heat sink. Investigate how sensitive the solutions are toward the choice of mesh size and/or element type.

3. How could one modify the current heat sink design to reduce the time for it to reach steady-state conditions? Demonstrate through a comparative FE analysis.

4. In practice, it's most likely that the heat flux at the base of the heat sink will vary as a function of time, say by increasing linearly from 0 to 1000 W m^{-2} over 200 sec. How can you model such a changing boundary condition in ABAQUS?

Plate Perforation Analysis

Type of Solver: ABAQUS CAE/Explicit

Dynamic Analysis – Plate Perforation from Projectile Impact

Introduction:

Perforation of metallic plates during projectile impact is a complex process, commonly involving elastic and plastic deformation, strain and strain rate hardening effects, thermal softening, crack formation, adiabatic shearing, plugging, petalling and even shattering. These effects depend on the properties and geometries of projectile and target and on the relative velocity of the colliding bodies. In this practical class, you will model the perforation of a deformable metallic plate when struck by a rigid, spherical projectile over a range of impact velocities. You will run analyses assuming both rate dependent and rate-independent material behaviour, and compare your predictions.

Problem Description:

You have been asked by the Ministry of Defence to determine the ballistic limit (minimum impact velocity required to fully perforate a target) of 0.4 mm thick steel plates, impacted by spherical projectiles that are 8 mm in diameter. The projectiles have a mass of 0.002 kg. The steel mechanical properties have been measured for you. The plastic mechanical behaviour is well described by the Johnson and Cook phenomenological plasticity model.

Johnson-Cook Theory

The Johnson & Cook phenomenological plasticity relation defines the flow stress as a function of equivalent plastic strain, strain rate and temperature. The model is frequently used in impact analyses because of its simplicity. One other major benefit of this model is that the various phenomena such as strain hardening, strain rate hardening and temperature softening can be uncoupled. When only strain hardening and temperature softening are coupled, the effective stress is given by:

(1) $$\bar{\sigma}_d = \left[A + B(\bar{\varepsilon}_{pl})^n \right](1 - \hat{\theta}^m)$$

When strain rate hardening is deemed to be significant, then the dynamic flow stress is expressed by the following relation:

(2) $$\bar{\sigma}_d = \left[A + B(\bar{\varepsilon}_{pl})^n \right]\left[1 + C \ln\left(\frac{\dot{\bar{\varepsilon}}_{pl}}{\dot{\varepsilon}_0} \right) \right](1 - \hat{\theta}^m)$$

where σ_d is the dynamic flow stress, $\bar{\varepsilon}_{pl}$ is the equivalent plastic strain, $\dot{\bar{\varepsilon}}_{pl}$ is the equivalent plastic strain rate, $\dot{\varepsilon}_0$ is a reference strain rate, *A*, *B*, *n*, *m* and *C* are material parameters and $\hat{\theta}^m$ is the non-dimensional temperature. The constant *A* is the yield stress under quasi-static conditions, *B* and *n* are strain hardening parameters, *m* controls the temperature dependence and *C* the strain rate dependence.

(a) Build a 3-dimensional FE model. Since the plates are so thin, assume their geometry can be represented by a shell with an 80 mm diameter. Assume also that the projectile does not deform (either elastically or plastically). Beyond the elastic limit, the strain hardening behaviour of the plate is best described using the Johnson and Cook constitutive plasticity model (Eq.1) – recall that this equation neglects any strain rate hardening contribution. Fracture in the plates will be modelled using a very simple critical plastic strain fracture criterion. The Johnson and Cook material property data for Weldox 460E steel are given in Table I. Using this information, determine the ballistic limit of the steel plates when struck normal to their plane, using a measured (quasi-static) uniaxial plastic fracture strain of 0.33. Predict also the energy absorbed by the steel plates for 5 impact speeds between the ballistic limit and 600 m s^{-1}, when struck normal to their plane.

(b) Strain rate hardening effects were neglected in **(a)**. However, the steel plates are known to exhibit rate-hardening behaviour. Fortunately, the Johnson and Cook plasticity model can be modified to include the effects of strain rate hardening (Eq.2). Modify your existing material model to include the Johnson and Cook strain rate hardening parameters (Table I) and the strain rate-dependent fracture behaviour (Table II). Re-run your simulations, and predict new ballistic limit and the absorbed energy as a function of the impact speed once more. Compare these predictions with those from **(a)**. Comment on the significance of strain rate hardening in this particular steel alloy.

(c) (OPTIONAL) Determine the ballistic limit of the steel plates when impacted at 45°. Use the fully coupled form of the Johnson and Cook equation.

Material Property Data

Table I   Johnson and Cook Material Property Data

A	B	n	m	T_{melt}	T_{tran}	C	Ref.Strain rate
Mpa	Mpa			K	K		s^{-1}
310	1000	0.65	1	1673	293	0.07	0.01

Table II   Rate-dependent Fracture Data

Strain Rate s^{-1}	Stress Triaxiality	Fracture Strain
0.01	0.33	0.33
0.1	0.33	0.32
1	0.33	0.31
10	0.33	0.30
100	0.33	0.29
1000	0.33	0.28
10000	0.33	0.27

Table III

| Density	Elastic Constants	
ρ	E	ν
kg m^{-3}	GPa	
7800	200	0.33

Solution (a):

- Start ABAQUS/CAE. At the **Start Session** dialogue box, click **Create Model Database.**

- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialogue box appears; name the model `Plate Perforation`


A. MODULE→PART

Under the Part module, we will construct the plate and projectile.

1. From the main menu bar, select **Part→Create**

2. The **Create Part** Dialogue box appears. Name the part `Steel Plate` and fill in the options as shown in **Fig.A1**. Click Continue to create the part.

3. From the main menu bar, select **Add→Circle**

    (a) Select the co-ordinates (*0, 0)* for the centre of the circle in the prompt area**.**

    (b) Select the co-ordinates (0.04*, 0)* for the perimeter point

    (c) Click [X] and then **Done** in the prompt area.

**4.** You must now create two partitions. From the main menu bar
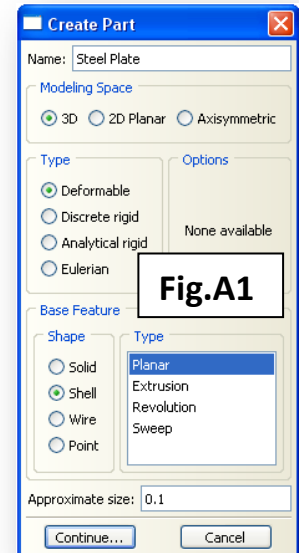
select **Tools→Partition**. The **Create Partition** dialogue box will open (**Fig.A2**).

    (a) Select **Face** as the **Type** and **Sketch** as the **Method**.

    (b) Using the mouse cursor select the edge of the part.

    (c) Using the **Create Circle** tool, create two circles; one with a diameter of 70 mm and the other with a diameter of 10 mm

    (d) Click [X] and then **Done** in the prompt area.

Fig.A1

Fig.A2
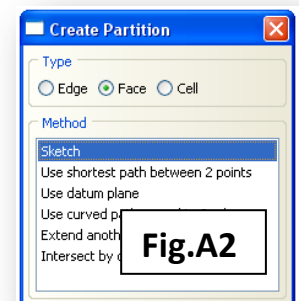
5. Now create an **Analytical Rigid** spherical projectile with an 8 mm diameter (**Fig.A3**). Once done, create a **Reference Point** at the centre. (**Tools→Reference Point**).

6. Create a set at the **Reference Point** and call it `Projectile Set.`

Fig.A3

Reference Point
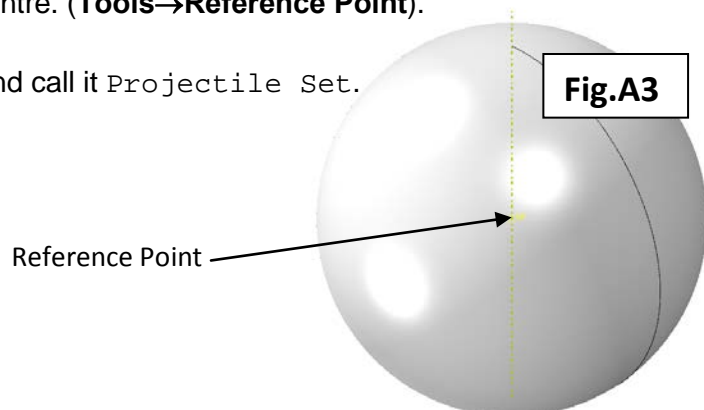
B. MODULE→PROPERTY

In this module (property), you will define the plate material properties. The strain hardening behaviour will be described using the Johnson and Cook plasticity relation. Fracture will be modelled by defining a critical plastic strain.

1. From the toolbox, select the **Create Material** tool. The Edit Material dialogue box will open.

   (a) Name the material `Weldox 460E`.

    (b) Define the density and elastic constants (Table III).

    (c) Select **Mechanical→Plasticity→Plastic** as shown in **Fig.B1**.

    (d) Under **Hardening**, select **Johnson-Cook** (**Fig.B2**). Fill in the data according to Table I.

    (e) To define the critical plastic strain at fracture, select **Mechanical→Damage for Ductile Metals→Ductile Damage** (**Fig.B3**). The quasi-static fracture strain is 0.33. The stress triaxiality is 0.33 and the reference strain rate is 0.01.

    (f) In the **Suboptions** drop-down menu, select **Damage Evolution** (**Fig.B4**). Choose a **Displacement at Failure** value of 0.0001 (**Fig.B5**). Click **OK**. Click **OK** again.

Fig.B1

Fig.B2

Fig.B3

Fig.B4

Fig.B5
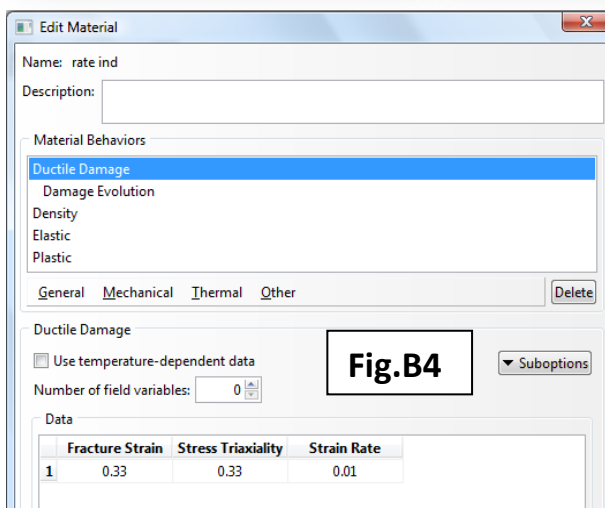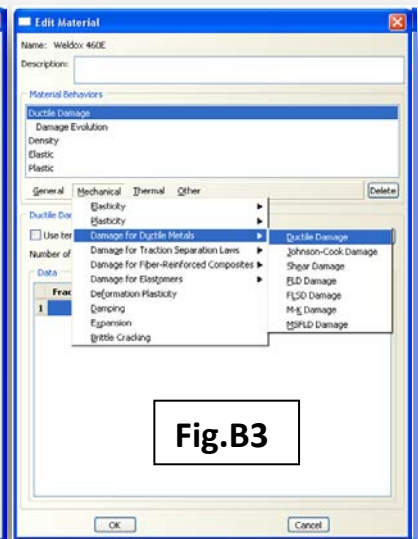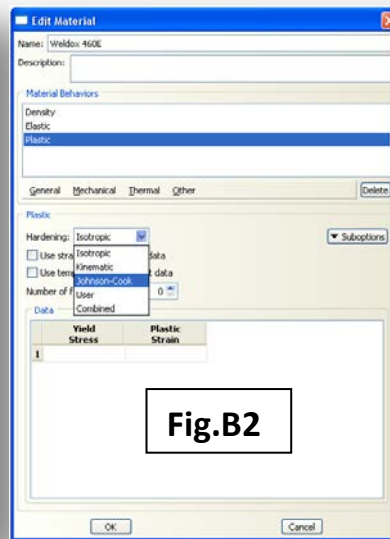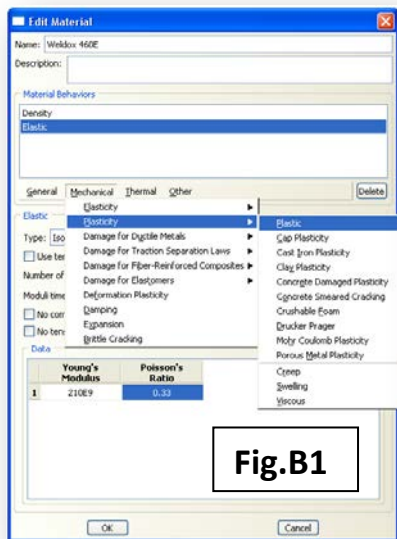
2. From the main menu bar select **Section→Create**. The **Create Section** dialogue box will open. Fill in the **Create Section** dialogue box as shown in **Fig.B6** and click **Continue**.

3. The **Edit Section** dialogue box will appear (**Fig.B7**). You must now define the thickness of the plate (0.4 mm). Click **OK**.

4. From the main menu bar select **Assign→Section**. Using the mouse cursor, click and drag across the plate and click **Done** in the prompt area. Click **OK** in the **Edit Section Assignment** dialogue box. Click **Done** in the prompt area.
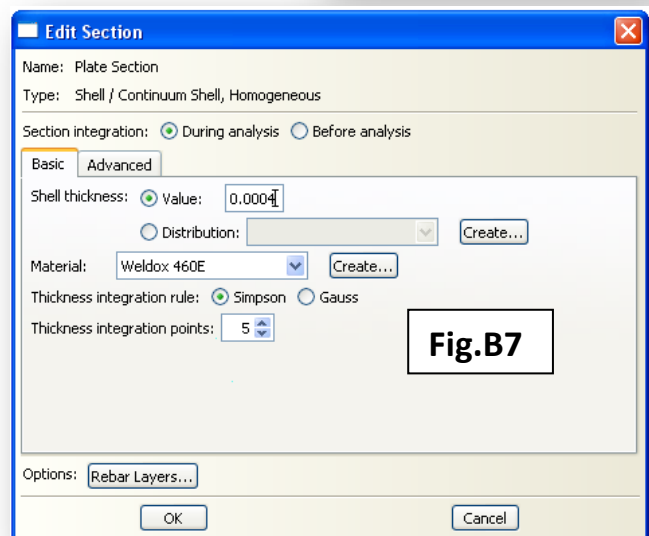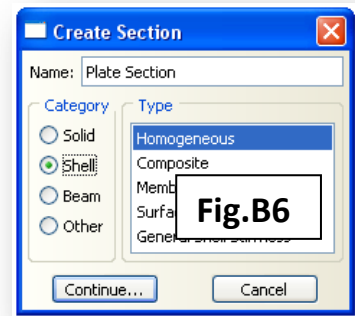
**Whilst in the Property module, you must also specify the projectile mass!!

5. From the main menu bar, select **Special→Inertia→Create**. The **Create Inertia** dialogue box will open. Name the mass condition `Projectile Mass` and select **Point Mass** as the **Type**. Click **Continue**. Select the **Reference Point** you created in the **Part** module as the point to which you wish to assign mass. Click **Done** in the prompt area and the **Edit Inertia** dialogue box will open. Select a mass of 0.002 kg and click **OK**.

C. MODULE→ASSEMBLY

1. From the main menu bar select **Instance→Create**. The **Create Instance** dialogue box will open. Select both of the parts and choose **Independent** as the **Instance Type**. Click **OK**.

2. Translate the projectile in the **Positive Z** direction by 6 mm (**Instance→Translate**). Click **OK** in the prompt area.


D. MODULE→STEP

---

1. From the main menu bar, select **Step→Create**. The **Create Step** dialogue box will appear Name the step `Impact Step`.

2. Select **General** from the **Procedure Type** options.

3. Select **Dynamic, Explicit** from the list of analysis types. Click **Continue**. The **Edit Step** dialogue box will open. Choose a time period of 0.0002 s (200 µs). Toggle on **NLGEOM**. Click **OK**.

4. Failed elements will need to be removed from the mesh in order to monitor crack propagation in your analyses. In the **Field Output Requests**, toggle on **Status**, which can be found under the **State/Field/User/Time** options.

5. To monitor the projectile velocity, create a new **History Output Request** called `Projectile Velocity`. Choose `Projectile Set` for the **Domain** and select V3, from the V options underneath the **Displacement/Velocity/Acceleration** list.

E. MODULE→INTERACTION

---

1. From the main menu bar select **Interaction→Create**. The **Create Interaction** dialogue box will open.

    (a) Name the interaction `Projectile Plate Contact`.

    (b) Choose **Surface-to-Surface** for the **Types for Selected Step**.

    (c) Click **Continue.**

2. You will be prompted to select the first (**Master**) surface. Using the mouse cursor, select the surface of the analytical rigid projectile and click **Done** in the prompt area.

3. When prompted, choose **Brown** in the prompt area, which represents the outer surface of the projectile.

4. You now need to define the second (**Slave**) surface involved in the contact. Select **Surface** from the prompt area. Using the mouse cursor, select the inner, partitioned region of the plate (**Fig.E1**). Click **Done** in the prompt area. Select **Brown** when prompted to choose a side for the shell surface.

5. The Edit Interaction dialogue box will open (**Fig.E2**). From the **Discretisation Method** select **Surface to Surface**.


Fig.E2

Fig.E1
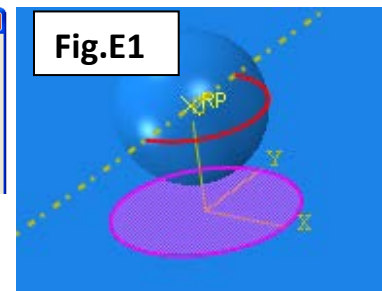
6. Click the **Create** tab next to **Contact Interaction Property**. The **Create Interaction Property** dialogue box will open.

7. Name it `Contact Interaction` and select **Contact** as the **Type**. Click **Continue** and the **Edit Contact Property** box will open. Select **Mechanical→Normal**, and accept the default options by clicking **OK**. Click **OK** again.

F. MODULE→LOAD
---

1. From the main menu bar select **BC→Create**. The **Create Boundary Condition** dialogue box will open (**Fig.F1**)

     (a) Name the boundary condition `Clamp`.

     (b) Choose **Mechanical** for the category.

     (c) Choose **Symmetry/Antisymmetry/Encastre** for the **Types for Selected Step**.

     (d) Click **Continue**.

     (e) You must now select the region for the **Clamp** boundary condition as shown in **Fig.F2**.


Fig.F1

Fig.F2
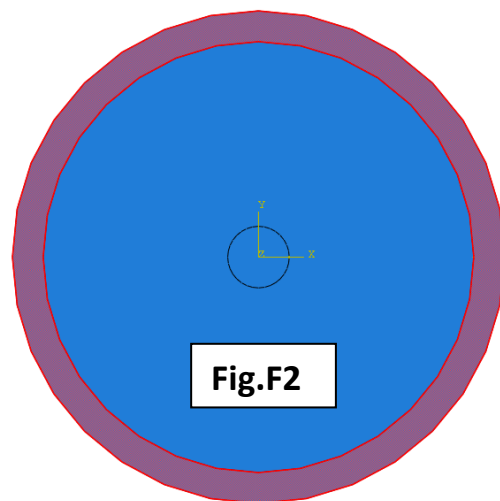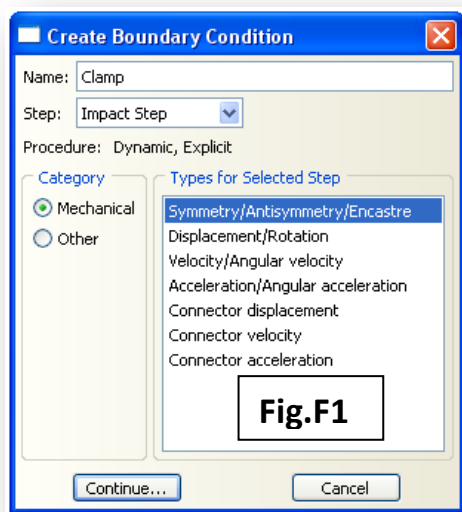
2. Click **Done** in the prompt area. The **Edit Boundary Condition** dialogue box will open. Select **Encastre** and click **OK**.

3. Create a further boundary condition that will allow the projectile to move in the **Z-direction (axis-3)** only. Assign this boundary condition to the projectile **Reference Point** that you created earlier.

4. To define the projectile velocity, from the main menu bar select **Predefined Field→Create**. Select the options shown in **Fig.F3** for the **Create Predefined Field** dialogue box. Click **Continue**.

Fig.F3

5. When prompted to select the region for the **Predefined Field**, select the projectile **Reference Point** once more. Click **Done** in the prompt area.

6. In the **Edit Predefined Field** dialogue box that appears, choose V1=V2=0 and V3=-150. Click **OK** when done.

G. MODULE→MESH

---
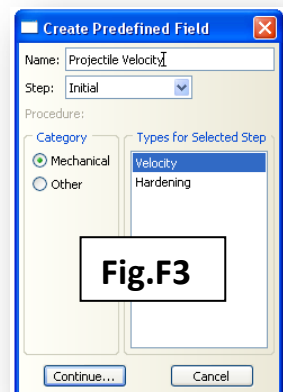
1. From the main menu bar, select **Mesh→Controls**. Using the mouse cursor, click and drag across the plate (remove the projectile using the viewing options!).

      (a) Select **Quad** as the **Element Type**.

      (b) Select **Free** as the **Technique**.

      (c) Select **Medial Axis** as the **Algorithm** type.

      (d) Click **OK**.

2. Using the **Seed** options, seed all 3 edges with 50 seeds using the **Seed Edge by Number** option.

3. From the main menu bar select **Mesh→Element Type**. Click and drag the mouse cursor over the plate and click **Done** in the prompt area. The **Element Type** dialogue box will open. Select **Explicit** from the **Element Library** and **Shell** from the **Element Family** options. Choose **Quad** elements and click **OK** to choose the **S4R** element type. Click **Done** in the prompt area.

4. From the main menu bar select **Mesh→Instance.** Using the mouse cursor, select the plate and click **Done** in the prompt area

**There is no time during the session today to conduct a mesh sensitivity analysis. However, you should bear in mind that, for simulations of this type, the results can be very sensitive to your choice of mesh. Under normal circumstances, you _MUST_ conduct a sensitivity analysis.

G. MODULE→JOB

---

1. Create a job titled `WeldoxBL` and submit the job for analysis. The job should take approximately one to two minutes.

H. MODULE→VISUALISATION

---

1. From the main menu bar select **Results→Field Output**. From the **Field Output** dialogue box select **Mises Stress**. Make a note of whether or not the plate has fully perforated (i.e. **Fig.H1**). If not, increase the impact velocity in the **Load** module. If it has, decrease the impact velocity in the **Load** module. Continue to do this until you converge upon the ballistic limit.

2. The projectile velocity data can be found from the **History Output** options accessible from the main menu bar. Using this information, calculate the absorbed energy (energy lost by the projectile) for impact velocities of 150, 200, 300, 400 500 and 600 m s^{-1}.
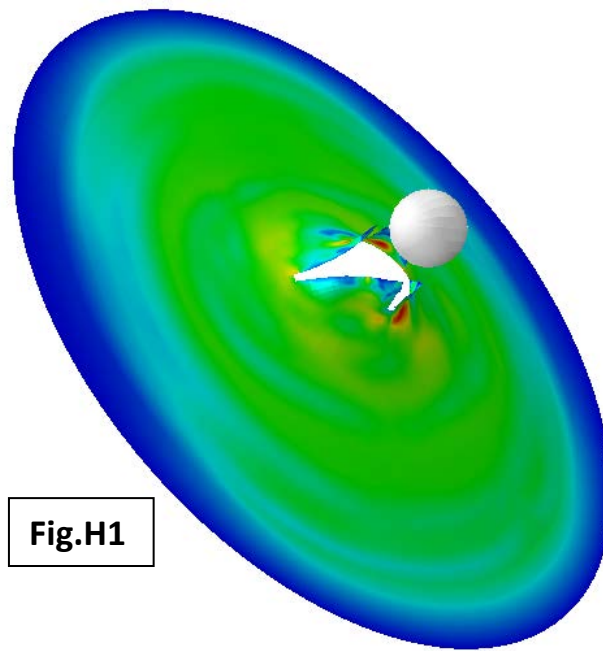
Fig.H1

Solution (b):

- Return to the **Property** module. Using the strain rate hardening and strain rate fracture data from Table I and Table II, define strain rate-hardening behaviour in your material model

A. MODULE→PROPERTY

1. From the main menu bar select **Material→Edit→Weldox 460E**.

   (a) Select **Mechanical→Plasticity→Plastic**.

   (b) Under the **Suboptions** tab select **Rate Dependent**.

   (c) Under **Hardening** select **Johnson-Cook** and fill in the rate-hardening data from Table I

2. Select **Ductile Damage** from the **Material Behaviours** box. Fill in the rate-dependent fracture strain data as shown in Table II.

B. MODULE→JOB

1. Determine the ballistic limit of the plate with these new material property characteristics by submitting the necessary jobs. *You may need to alter the step time for low impact velocities. Submit 5 further jobs at impact speeds between the ballistic limit and 600 m s^{-1}.

C. MODULE→VISUALISATION

1. View the **Mises Stress** contours by selecting **Results→Field Output** from the main menu bar. Calculate the absorbed energy by extracting the projectile velocity history.

OPTIONAL QUESTIONS

---

1. Determine the ballistic limit of the plates when struck at an angle of 45°.

2. The effects of friction have not been considered in these analyses. Using the FE model, and assuming a friction co-efficient between the projectile and the plate of 0.18, determine whether or not there is a contribution to absorbed energy from friction. How much more significant is friction when the angle of incidence is 45° compared to when the plate is struck at normal incidence for an impact velocity of 300 m s^{-1}. **(*Hint; return to your contact interaction property and search under tangential behaviour.)**

3. The strain and strain rate-dependent behaviour has been described using the Johnson-Cook constitutive relation. Is there an alternative way of defining this data in the material editor?

4. In the strain rate-dependent simulations, you defined rate-dependent fracture strains for strain rates as high as 10000 s^{-1}. Is this strain rate-dependent fracture data sufficient to cover the range of strain rates that are generated at the highest impact velocities? **(*Hint; you will need to edit the field output requests to include strain rates as output data.)**

Fully-Coupled Thermo-Mechanical Analysis

Type of solver: ABAQUS CAE/Standard

Adapted from: ABAQUS Example Problems Manual

Extrusion of a Cylindrical Aluminium Bar with Frictional Heat Generation

Problem Description:

The figure below shows the cross-sectional view of an aluminium cylindrical bar placed within an extrusion die. The bar has an initial radius of 100 mm and a length of 300 mm, and its radius is to be reduced by 33% through an extrusion process. The die can be assumed as an isothermal rigid body. During the extrusion process, the bar is forced downwards by 250 mm at a constant displacement rate of 25 mm s^{-1}. The generation of heat attributable to plastic dissipation inside the bar and the frictional heat generation at the die-workpiece interface causes temperature of the workpiece to rise. When extrusion is completed, the workpiece is allowed to cool in the ambient air. The ambient surrounding is at 20 °C, with a coefficient of heat transfer of 10 W m^{-2} K^{-1}.

Formulate an axisymmetric FE model to predict (i) the geometry of the deformed bar, (ii) the plastic strain distribution and (iii) the temperature evolution, at various stages of the extrusion process.
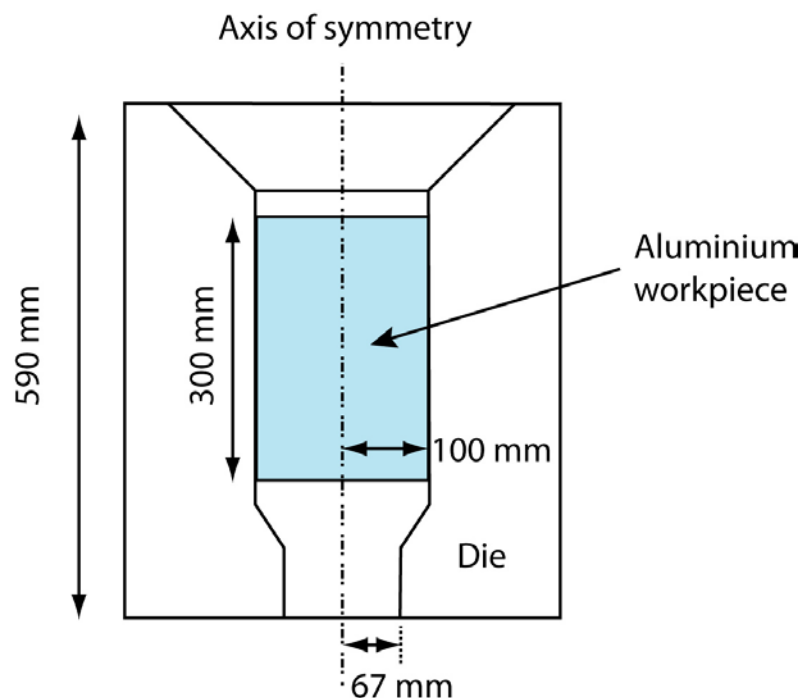
SOLUTION:

- Start ABAQUS/CAE. At the **Start Session** dialog box, click **Create Model Database**.

- From the main menu bar, select **Model→Create**. The **Edit Model Attributes** dialog box appears, name the model TM_Coupled


A. MODULE → PART

We will construct an axisymmetric model consisting of a deformable workpiece and a rigid die.

(a) To sketch the aluminium alloy workpiece

1. From the main menu bar, select **Part→Create**

2. Name the part Workpiece. Use the settings are shown in **Fig.A1**. Ensure that the **Modeling Space** is set to **Axisymmetric** and the **Type** as **Deformable**.

3. Sketch the Workpiece, the 4 vertices as shown in **Fig.A2** are (0,0), (0.1,0), (0,0.3) and (0.1,0.3) in metres.

   Note: When building an axisymmetric model, it is important to observe the position of various parts in relation to the axis of symmetry.
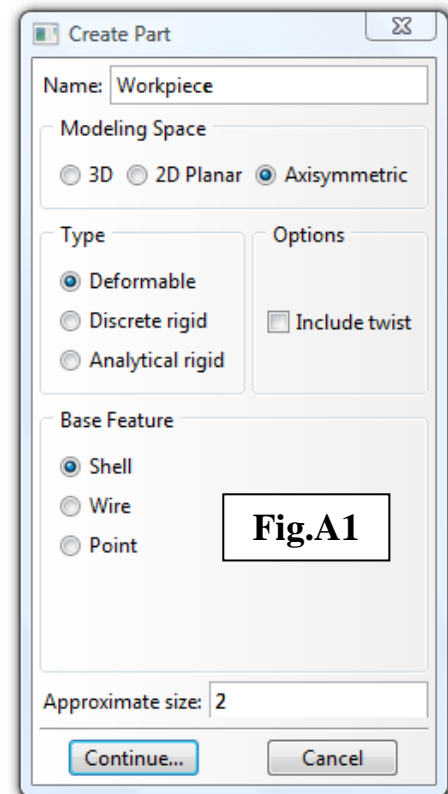
Fig.A1
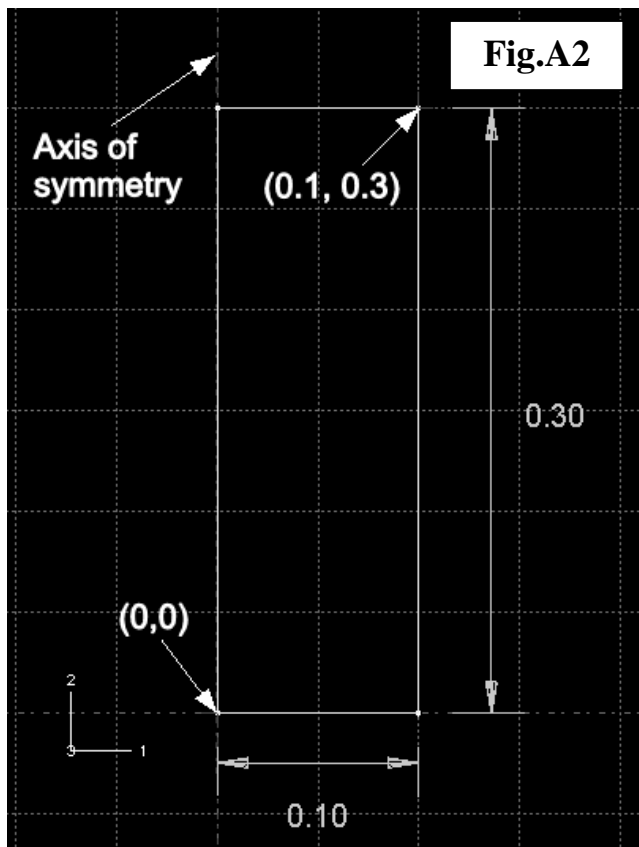
Fig.A2

(b) To sketch the rigid die

1. From the main menu bar, select **Part→Create**

2. Name the part Die. Apart from the name, all the other settings are the same as in **Fig.A1**.

   Note: Although the die is meant to be a rigid body in this analysis, here we choose to first

   build it as a deformable body and later apply a **Rigid body** constraint (Section E (c)).

3. Sketch the Die using the vertices given in **Fig.A3**.

   Note: Observe that all coordinates are followed correctly, so that the assembly of the

   workpiece and die can be carried out correctly later.

4. We also need to add a reference point to the die part to be used in rigid body constraint.

   From the main menu bar, select **Tools→Reference Point**. Pick point (0.067, -0.18) as

   denoted in **Fig.A3**, note that a yellow **RP** symbol appears.
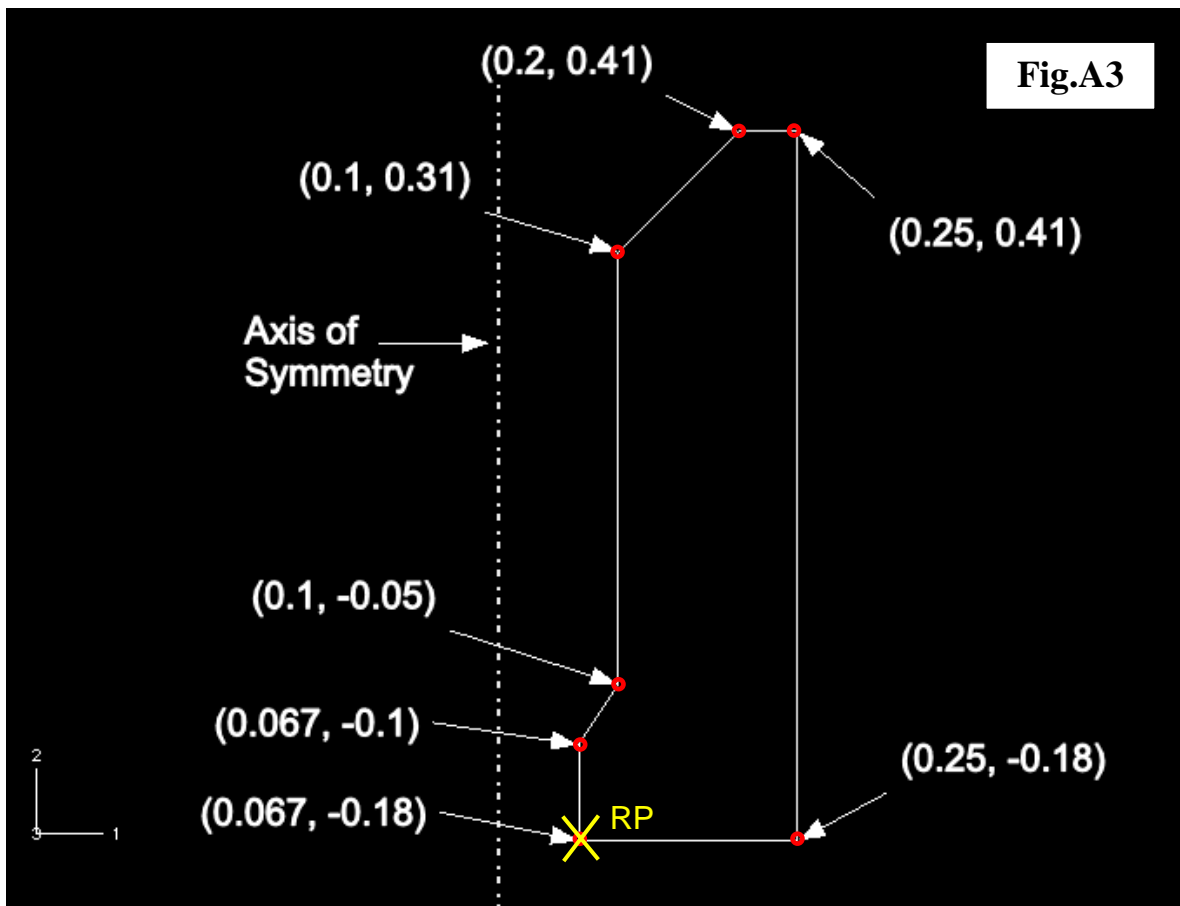

B. MODULE → PROPERTY

(a) To enter material properties of the workpiece:-

1. From the main menu bar, select **Material→Create**

2. Name the material `Aluminium`.

3. Create the following material properties:

   (i) **General→Density**   `2700` (kg m^{-3})

   (ii) **Thermal→Conductivity**

   - Under **Type** choose **Isotropic**

   - Toggle on **Use temperature-dependent data**
     (NB. Conductivity in W m^{-1} K^{-1}, Temp in °C)
     and use data shown in **Fig.B1**.
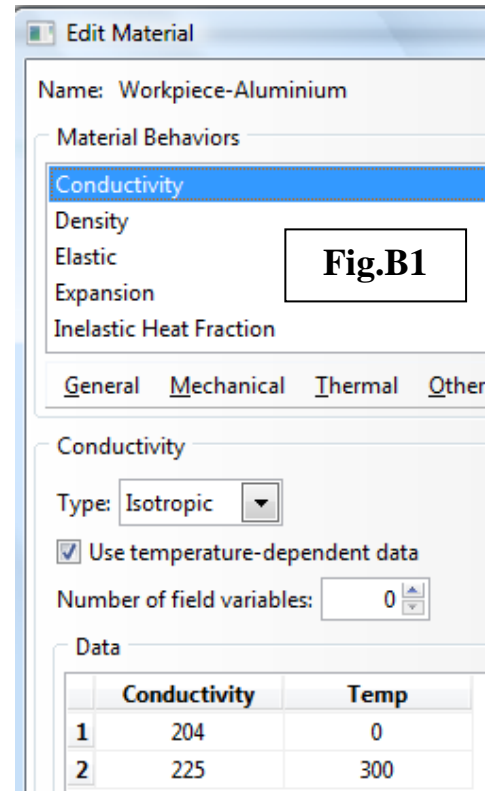
   (iii) **Thermal→Inelastic Heat Fraction**   `0.9`

   (iv) **Thermal→Specific Heat**   `880` (J kg^{-1} K^{-1})

   (v) **Mechanical→Elastic**

       Under **Type** choose **Isotropic**

       Young's Modulus: `69×10`^{9} (Pa)

       Poisson's Ratio: `0.33`

   (vi) **Mechanical→Expansion**

       Under **Type** choose **Isotropic**

       Reference temperature: `20` (°C)

       Expansion Coeff alpha: `8.42×10`$^{-5}$ (K^{-1})

Edit Material

Name: Workpiece-Aluminium

Material Behaviors

Conductivity
Density
Elastic
Expansion
Inelastic Heat Fraction

Fig.B1

<u>G</u>eneral   <u>M</u>echanical   <u>T</u>hermal   Other

Conductivity

Type: Isotropic ▼

☑ Use temperature-dependent data

Number of field variables: 0 ⬍

Data

	Conductivity	Temp
1	204	0
2	225	300

(vii) **Mechanical→Plasticity→Plastic**

Under **Hardening** choose **Isotropic** (**Fig.B2**)

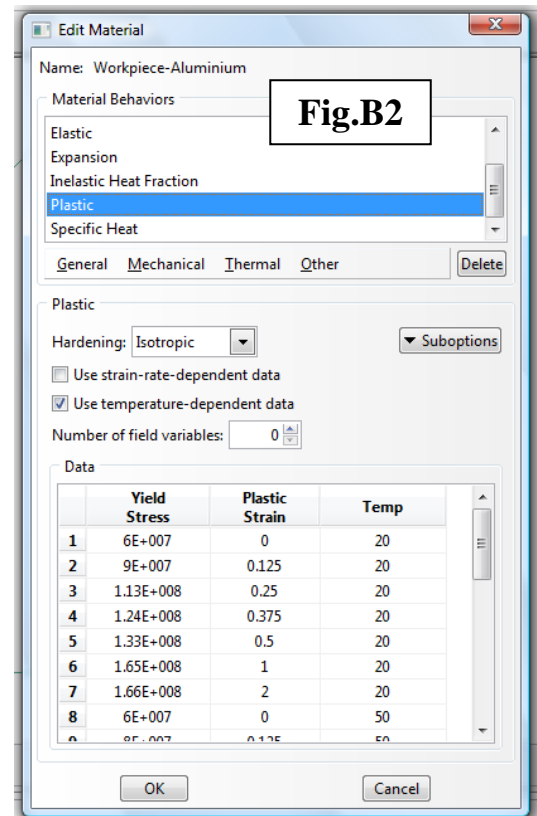Toggle on **Use Temperature-dependent data**

The complete set of data is given in **Table 1**.

(*Note:* The list of data can also be directly imported into ABAQUS/CAE if an ASCII text file is available (*NOT provided in this exercise*). To do this, right click within the table and choose **Read from File** 📂)

Table 1: Temperature-dependent flow stress of Aluminium

Yield stress	Plastic strain	Temp
6.00E+07	0	20
9.00E+07	0.125	20
1.13E+08	0.25	20
1.24E+08	0.375	20
1.33E+08	0.5	20
1.65E+08	1	20
1.66E+08	2	20
6.00E+07	0	50
8.00E+07	0.125	50
9.70E+07	0.25	50
1.10E+08	0.375	50
1.20E+08	0.5	50
1.50E+08	1	50
1.51E+08	2	50
5.00E+07	0	100
6.50E+07	0.125	100
8.15E+07	0.25	100
9.10E+07	0.375	100
1.00E+08	0.5	100
1.25E+08	1	100
1.26E+08	2	100
4.50E+07	0	150
6.30E+07	0.125	150
7.50E+07	0.25	150
8.90E+07	0.5	150
1.10E+08	1	150
1.11E+08	2	150

Fig.B2

Edit Material

Name: Workpiece-Aluminium

Material Behaviors

Elastic
Expansion
Inelastic Heat Fraction
Plastic
Specific Heat

General   Mechanical   Thermal   Other   Delete

Plastic

Hardening: Isotropic   ▼ Suboptions

☐ Use strain-rate-dependent data
☑ Use temperature-dependent data
Number of field variables: 0

Data

	Yield Stress	Plastic Strain	Temp
1	6E+007	0	20
2	9E+007	0.125	20
3	1.13E+008	0.25	20
4	1.24E+008	0.375	20
5	1.33E+008	0.5	20
6	1.65E+008	1	20
7	1.66E+008	2	20
8	6E+007	0	50
	9E+007	0.125	50

OK   Cancel

### (b) To enter material properties of the die:-

1. From the main menu bar, select **Material→Create**

2. Name the material `Die-Material`

4. Create the following material properties:

   Note: Since the die will be modelled as a rigid body and heat flow into the die is not modelled, the properties entered here will be inconsequential. However, non-zero values must be entered so that the ABAQUS/CAE solver can proceed.

   (i) **General→Density**   `2700` (kg m^{-3})

   (ii) **Thermal→Conductivity**   `200` (W m^{-1} K^{-1})

   (iii) **Thermal→Specific Heat**   `880` (J kg^{-1} K^{-1})

   (iv) **Mechanical→Expansion**

   Under **Type** choose **Isotropic**

   Reference temperature: `20` (°C)

   Expansion Coefficient - alpha: `8.42×10`$^{-5}$ (K^{-1})

   (v) **Mechanical→Elastic**

   Under **Type** choose **Isotropic**

   Young's Modulus: `200×10`^{9} (Pa)

   Poisson's Ratio: `0.3`

(c) To create the sections and assign them to the parts

1. From the main menu bar, select **Section→Create**

2. Name it `Section-workpiece` (**Fig.B3**). For **Category,** choose **Solid,** and set **Type** as **Homogeneous**.

Fig.B3

3. In the **Edit Section** dialogue box (**Fig.B4**), under **Material** pick `Aluminium`.

4. Now create a section for the die, call it `Section-die`.

5. Assign the sections to the relevant parts.

Fig.B4
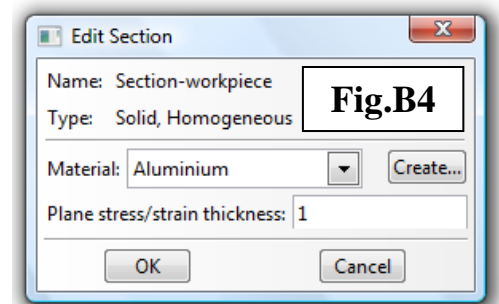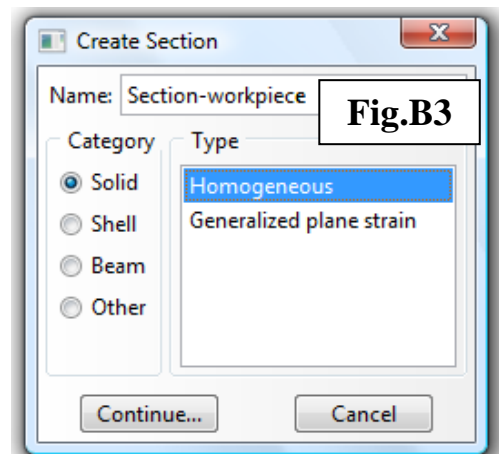
C. MODULE → ASSEMBLY

1. From the main menu bar, select **Instance→Create**

2. First create an instance of the `Die` part. Under **Instance Type**, make sure to select **Independent (mesh on instance)**. Toggle off **Auto-offset from other instances**.

3. Then create an instance of the `Workpiece` part, also as an independent instance. Make sure that **Auto-offset from other instances** is set as off, see **Fig.C1.**
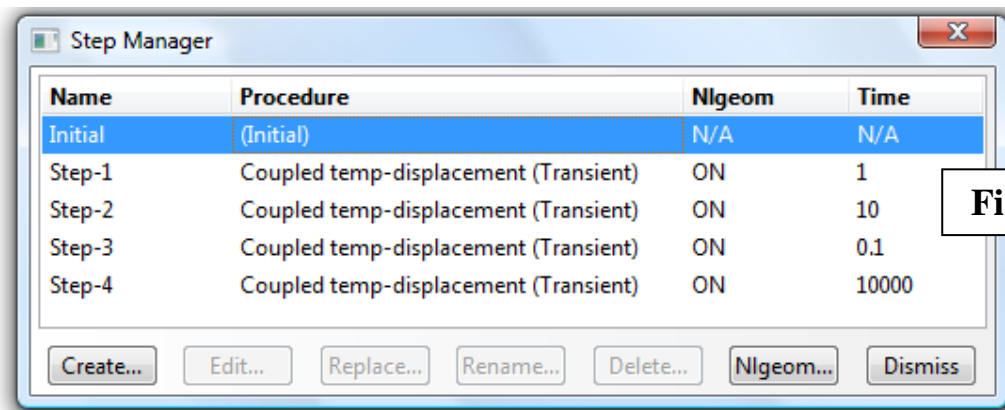
4. The complete assembly of the die and workpiece is depicted on the left panel of **Fig.C1**.
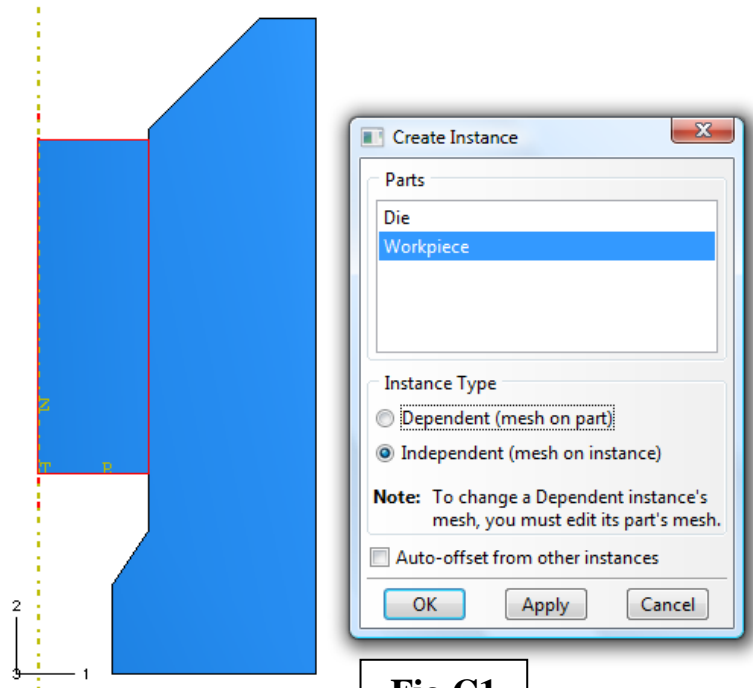
Fig.C1

D. MODULE → STEP

This fully-coupled thermal displacement transient analysis will consist of an initial step (exist by default) plus 4 additional steps (to be created in this section). **Fig.D1** shows the **Step Manager** dialogue box with all the steps correctly set up.

Fig.D1

Important: The "**Nlgeom**" option must be enabled to account for large strain plastic deformations.

(a) To create Step-1: Stabilise workpiece inside die

1. From the main menu bar, select **Step→Create**

2. Name it `Step-1` (**Fig.D2**). The **Procedure type** is **General→Coupled temp-displacement**

3. In **Edit Step** dialog box, under the **Basic** tab (**Fig.D3**), enter `Stabilise workpiece inside die` as the **Description**. To account for large plastic deformation, toggle on **Nlgeom**. To consider time-dependent plasticity, toggle on **Include creep/swelling/viscoelastic behavior**.

4. In **Edit Step** dialog box, click on the **Incrementation** tab (**Fig.D4**) and reduce the **Initial Increment size** to `0.1`. Toggle on **Max. allowable temperature change per increment** and enter `100.`

5. Accept the default settings under the **Other** tab.
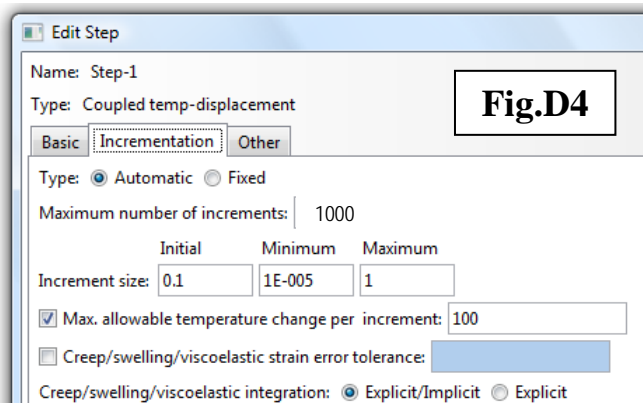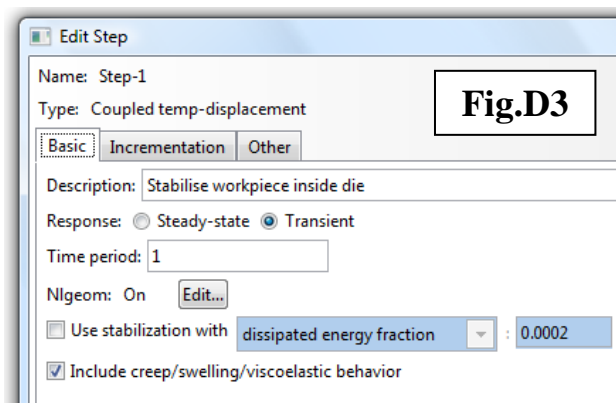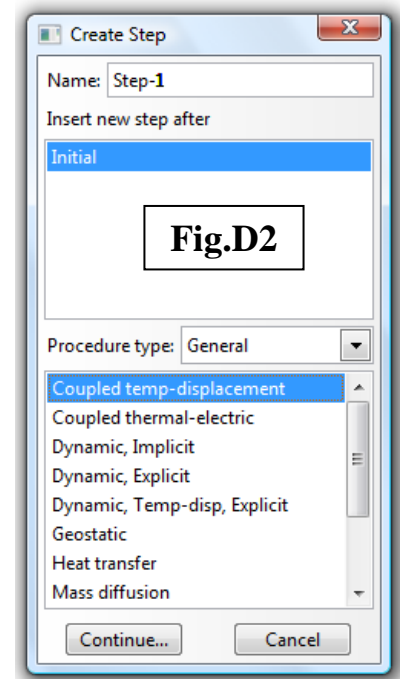
(b) To create Step-2: Extrusion

Create `Step-2`. As of `Step-1`, the **Procedure type** is **Coupled temp-displacement**. **Figs.D5** and **D6** show the parameters to be used.
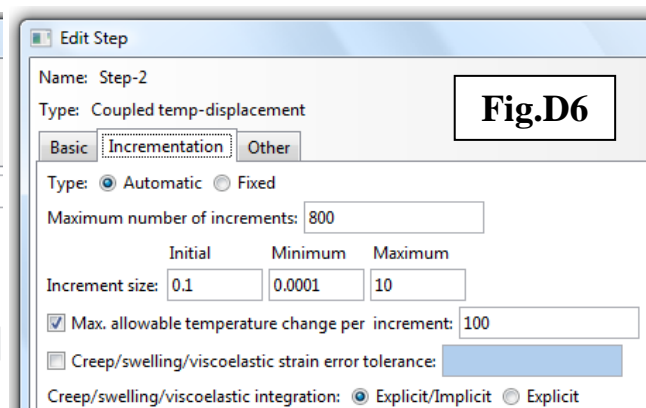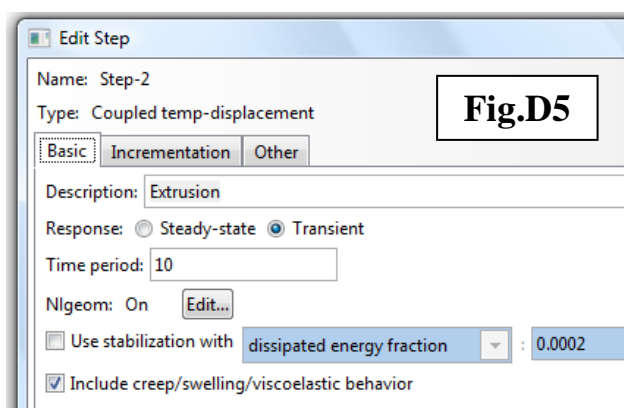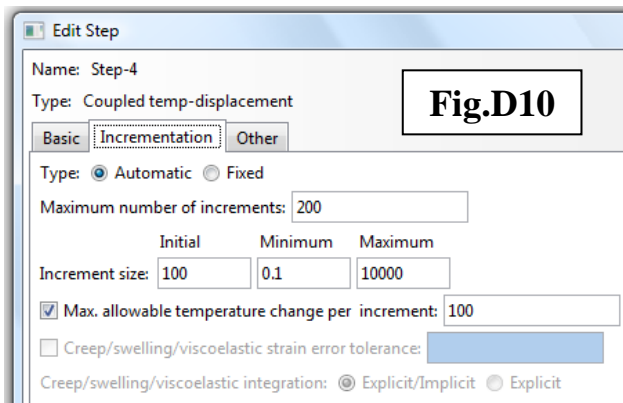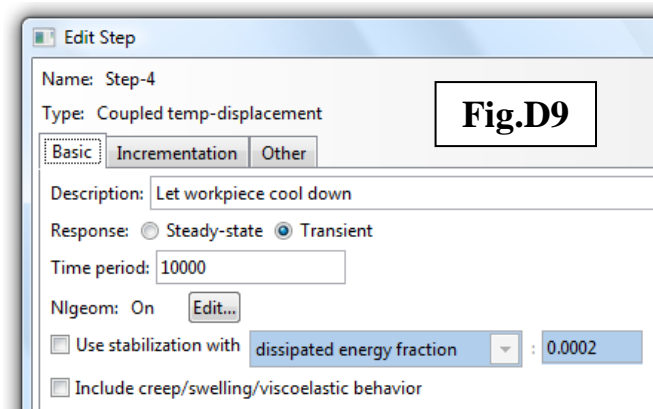
(c) To create Step-3: Remove contact pairs

Create `Step-3` and fill out the parameters as in **Figs.D7** and **D8**.

Fig.D7

Fig.D8
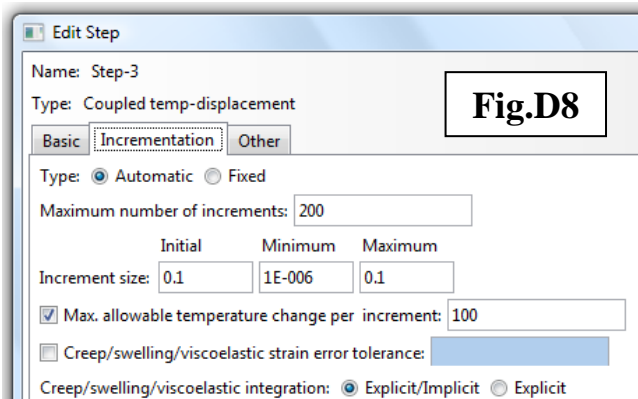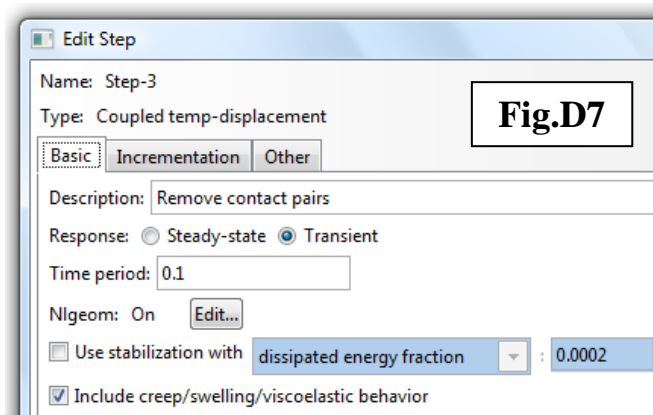
(d) To create Step-4: Let workpiece cool down

Create `Step-4` and fill out the parameters as in **Figs.D9** and **D10**.

Fig.D9

Fig.D10

E. MODULE → INTERACTION

(a) To create surfaces for interaction

1. From the main menu bar, select

   View→Assembly Display Options

2. Click the **Instance** tab, toggle off the visibility of
   `Workpiece-1`, see **Fig.E1**. Click **OK**.

3. From the main menu bar, select

   Tools→Surface→Create.

4. Name the surface `Surf-die-Contact`, pick the 5
   edges designated in **Fig.E2**. *Tip:* To make multiple
   selections, hold down the Ctrl-button while clicking.

5. Return to **Assembly Display Options** to toggle on the visibility
   of `Workpiece-1`, then toggle off `Die-1`. Click **OK**.

6. Create a surface called `Surf-workpiece-Vertical`, i.e.
   the vertical edge that comes into contact with the die, see
   Fig.E3.

7. Create a surface called `Surf-workpiece-Horizontal`,
   as designated in **Fig.E3**.

8. Finally, create another surface called
   `Surf-workpiece-Convect` that consists of 3 edges
   denoted in **Fig.E4**.

9. Toggle on both instances when finished assigning all surfaces.

Fig.E1

Fig.E2

Fig.E3

Fig.E4

84
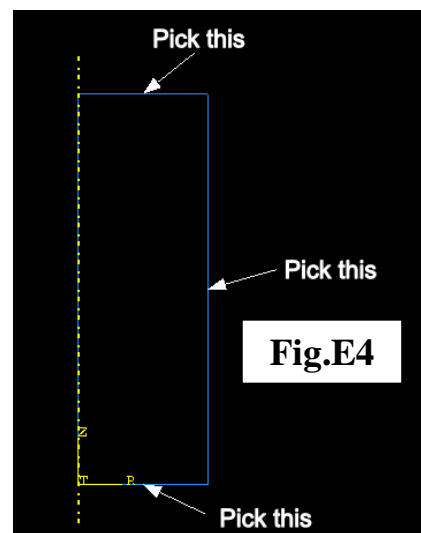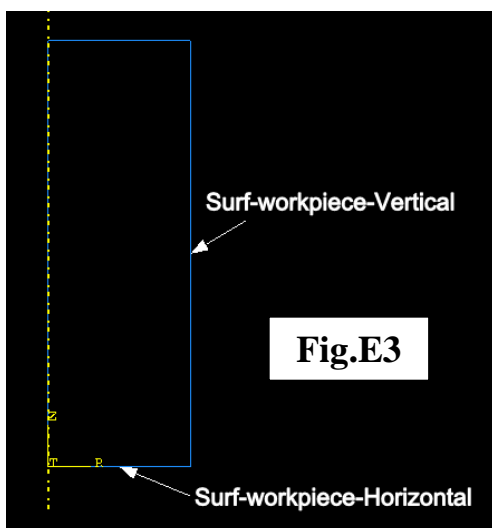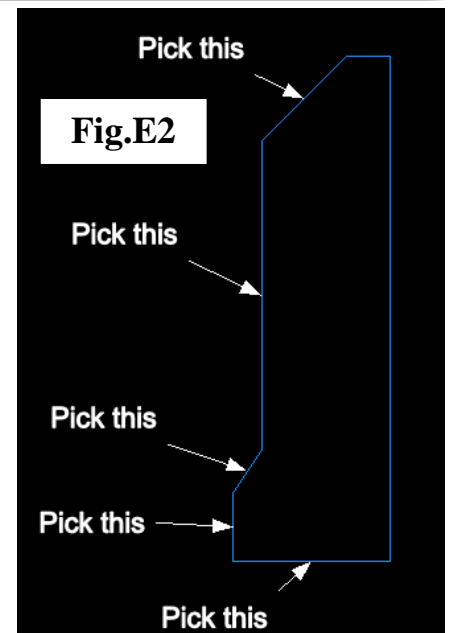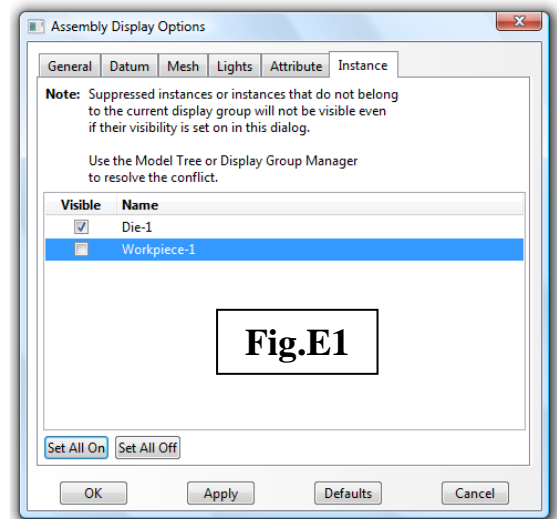
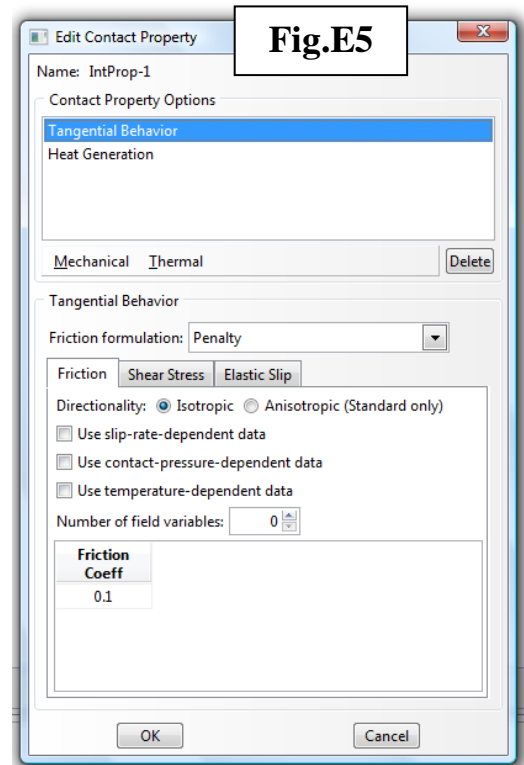### (b) To create the interaction property

1. From the main menu bar, select

   Interaction→Property→Create

2. Name it `IntProp-1`. Under **Type**, select **Contact**.

3. In the **Edit Contact Property** dialogue box (**Fig.E5**), add the following properties:

   (i) **Mechanical→Tangential Behavior**

   - For **Friction formulation**, choose **Penalty**

   - **Friction Coeff**: `0.1`

   (ii) **Thermal→Heat Generation**

   - Use `0.5, 0.5`

Fig.E5

(c) To create a rigid body constraint for the die

1. From the main menu bar, select **Constraint →Create**

2. Name the constraint `Die-RigidBody`. Under **Type**, select **Rigid body**.

3. The **Edit Constraint** dialogue box appears (**Fig.E6**). Under **Region type**, choose **Body(elements)** and then pick the die region.

4. For **Reference Point**, pick the yellow point denoted as **RP** (see **Fig.A3**), i.e. the reference point of the die.

5. Toggle on **Constraint selected regions to be isothermal**.

Fig.E6
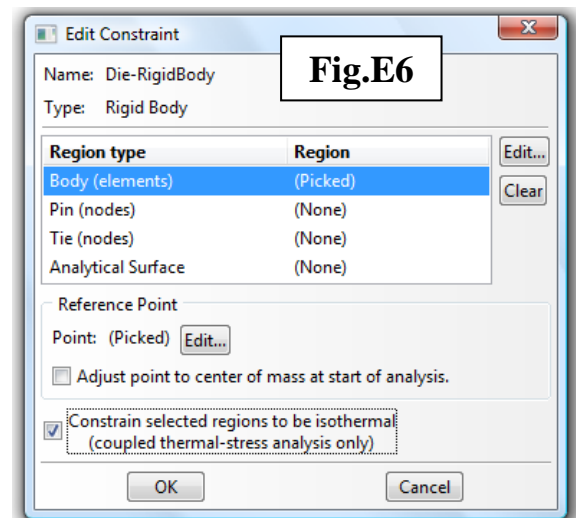
(d) To create the interactions

We will create 3 interactions, the end result is shown in **Fig.E7**. Note that not all interactions are active at all steps.
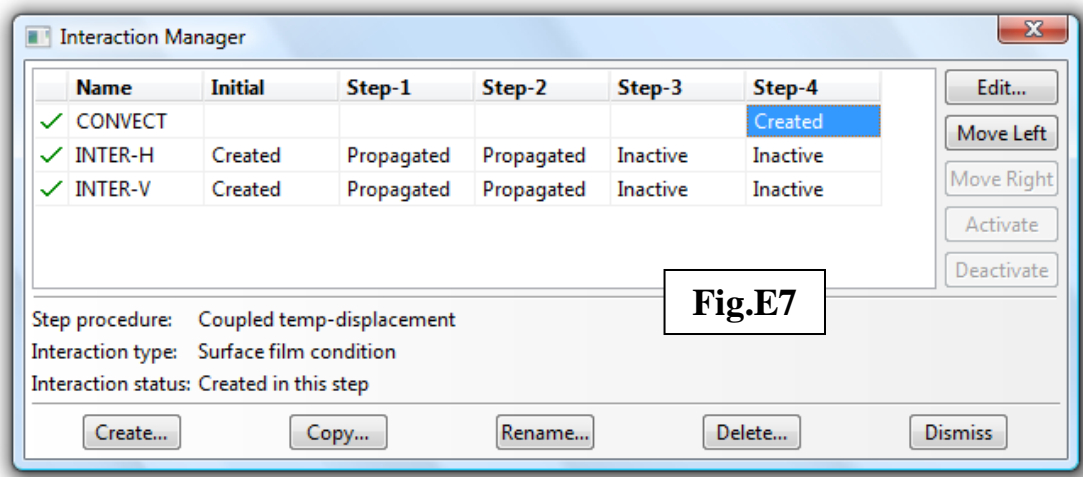
Fig.E7

- *Thermal film interaction*

1. From the main menu bar, select **Interaction → Create**

2. Name it CONVECT. Select Step-4 and **Surface film condition**, see **Fig.E8**.

3. To pick the surface, click on the Surfaces... button located at the right hand corner of the prompt area, then select Surface-workpiece-Convect.

4. In the **Edit Interaction** dialogue box, enter **Film coefficient** as 10 (W m^{-2} K^{-1}) and **Sink temperature** as 20 (°C).
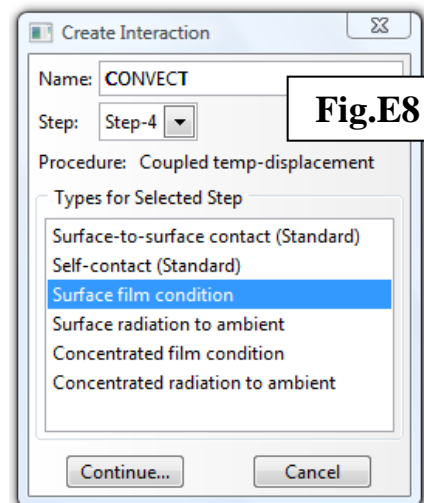
Fig.E8
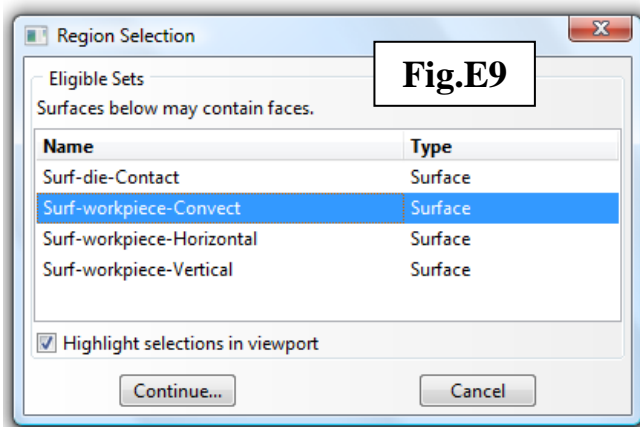
Fig.E9
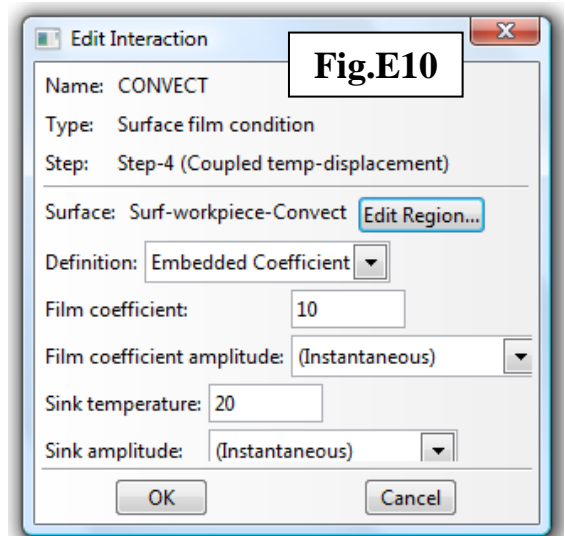
Fig.E10

- *Mechanical interactions*

1. From the main menu bar, select **Interaction →Create**

2. Name it `INTER-H`. Select **Step:** `Initial` and **Surface-to-surface contact (Standard)**, see **Fig.E11**.

3. For the master surface, select `Surf-die-Contact` (i.e. choose the stiffer of the pair)

4. Choose the slave type as **Surface**, then select `Surf-workpiece-Horizontal`.

5. The **Edit Interaction** dialogue box (**Fig.E12**) appears. Set **Degree of smoothing for master surface** as `0.48`. Accept the rest of the default settings. Note that the **Contact interaction property** is `IntProp-1` which was created earlier in *(a)*.

6. Now using similar procedures, create an interaction for `INTER-V`. Assign `Surf-die-Contact` as the master surface and `Surf-workpiece-Vertical` as the slave.

7. We also need to make `INTER-H` and `INTER-V` inactive during `Step-3` (*Remove contact pairs*) and `Step-4` (*Let workpiece cool down*). From the main menu bar, select **Interaction →Manager** to bring up the **Interaction Manager** dialogue box (**Fig.E7**). Click on the box that corresponds to `INTER-H` and `Step-3`, then click on **Deactivate**. Repeat for `INTER-V.`
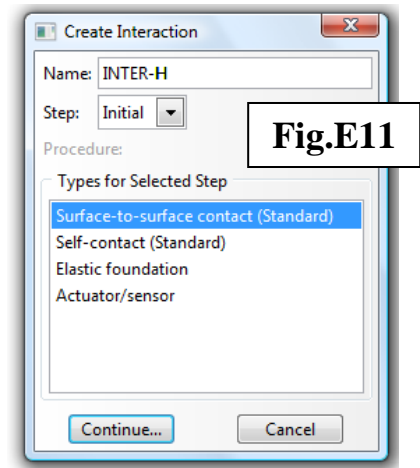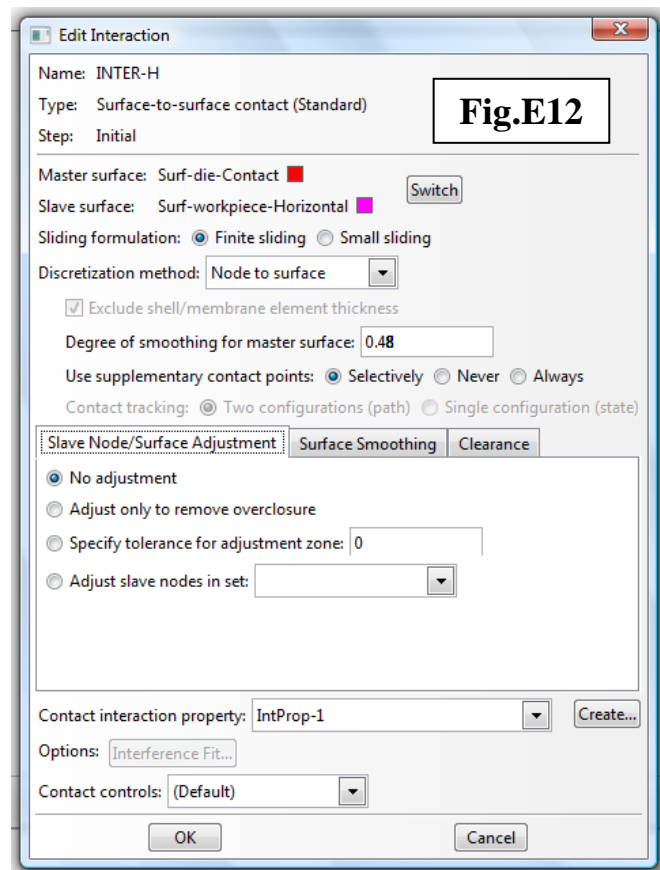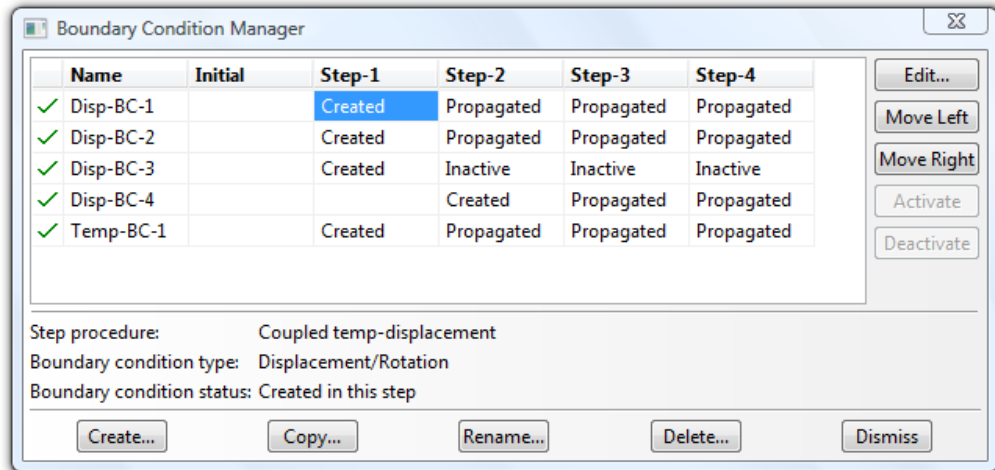

Fig.E11

Fig.E12

F. MODULE → LOAD


(a) To create the boundary conditions

- We will create 5 boundary conditions (BCs), the end result is shown in **Fig.F1**. Four of which are displacement boundary conditions (`Disp-BC`) and the last is a temperature boundary condition (`Temp-BC`). Note that not all BCs are active at all times.
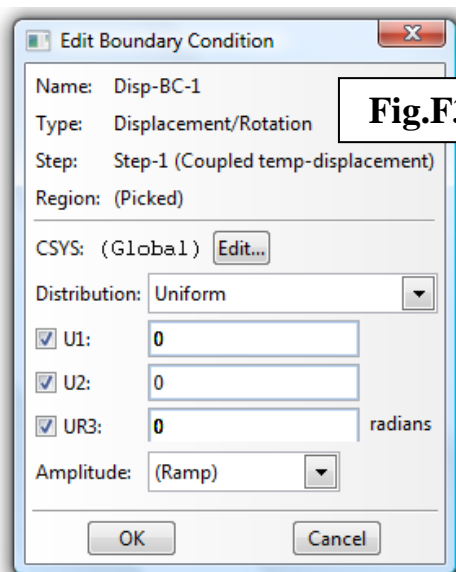
Fig.F1

(i) `Disp-BC-1`

1. From the main menu bar, select **BC→Create**

2. Name it `Disp-BC-1` (**Fig.F2**). Assign to **Step:** `Step-1`. **Category**: **Mechanical→Displacement/Rotation**

3. Pick **RP**, the reference point of the die.

4. Set **U1** = **U2** = **UR3** = 0 (To ensure that the die remains static throughout the simulation), see **Fig.F3**.

Fig.F2

Fig.F3

88
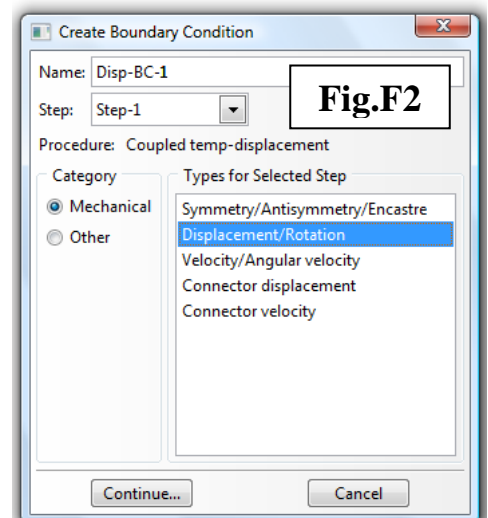
(ii) `Disp-BC-2`

1. From the main menu bar, select
   BC→Create

2. Name it `Disp-BC-2`. Assign to **Step:**
   `Step-1`. **Category**:
   Mechanical→Displacement/Rotation

3. Pick the edge corresponding to the axis
   of the workpiece, see **Fig.F4**.

4. Set **U1** = 0 (To ensure that the
   workpiece remains axisymmetric
   throughout the simulation).

Fig.F4

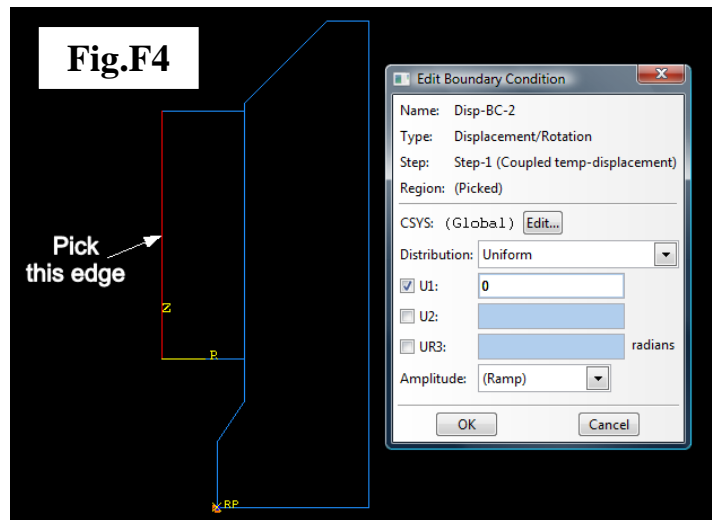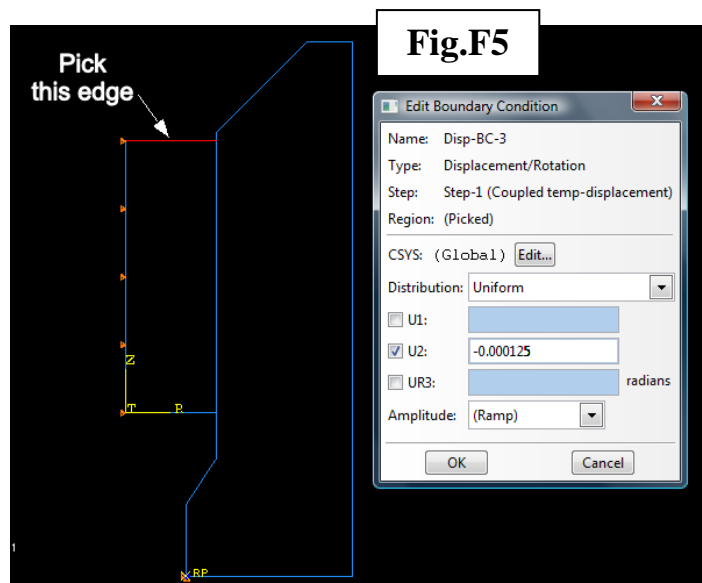## (iii) `Disp-BC-3`

1. From the main menu bar, select **BC→Create**

2. Name it `Disp-BC-3`. Assign to **Step:**
   `Step-1`. **Category**:
   Mechanical→Displacement/Rotation

3. Pick the edge corresponding to the top
   surface of the workpiece, see **Fig.F5**.

4. Set **U2** = `-0.000125` (m)
   Note: A relatively *small* vertical
   displacement is assigned to the top
   surface of the workpiece at the start of
   the simulation to establish contacts at the
   interfaces.

Fig.F5

5. Deactive this BC for `Step-2` and beyond, use the **Boundary Condition Manager** to do this -
   see **Fig.F1**.

(iv) `Disp-BC-4`

1. From the main menu bar, select
   BC→Create

2. Name it `Disp-BC-4`. Assign to **Step:**
   `Step-2`. **Category**:
   Mechanical→Displacement/Rotation

3. Pick the edge corresponding to the top
   surface of the workpiece, see **Fig.F6**.

4. Set **U2** = `-0.25` (Displace the workpiece
   by 250 mm downwards, i.e. to simulate
   the extrusion process)

Fig.F6
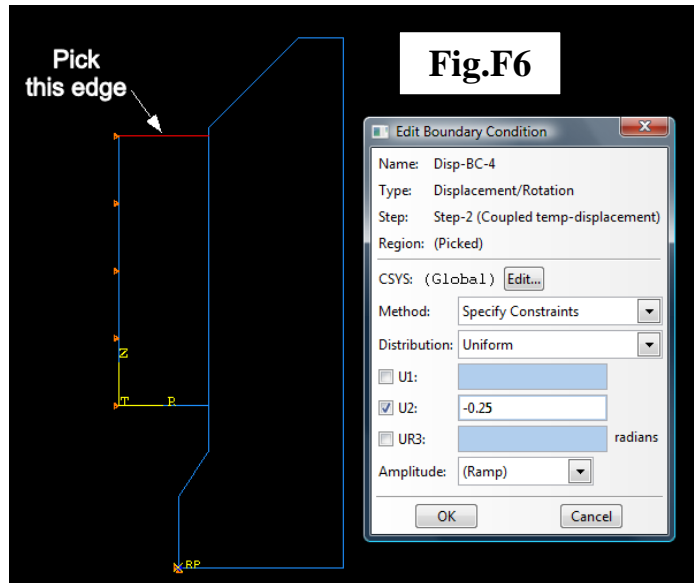
(v) `Temp-BC-1`

1. From the main menu bar, select
   BC→Create

2. Name it `Temp-BC-1`. Assign to **Step:**
   `Step-1`. **Category:**
   Other→Temperature

3. Pick the RP on the die, see **Fig.F7**.

4. Set the Magnitude as `20` (°C). The
   temperature of the die remains constant
   throughout simulation since we are not accounting for heat transfer into the die.

Fig.F7
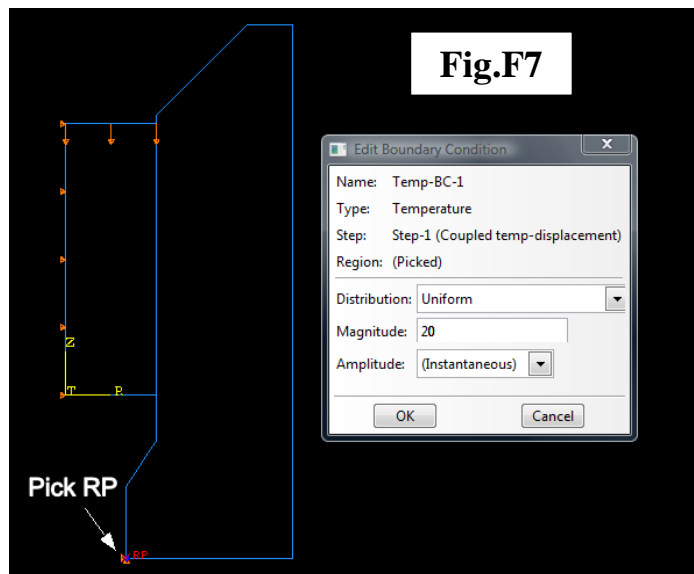
### (b) To create a thermal field for the workpiece

1. From the main menu bar, select **Predefined Field→Create**

2. Name it `Field-workpiece`. Assign to **Step:** `Initial`. **Category: Other→Temperature**

3. Pick the `Workpiece` instance. In the **Edit Field** dialogue box, enter `20` (°C) as the
   Magnitude. This is the initial temperature, the temperature in subsequent steps will be
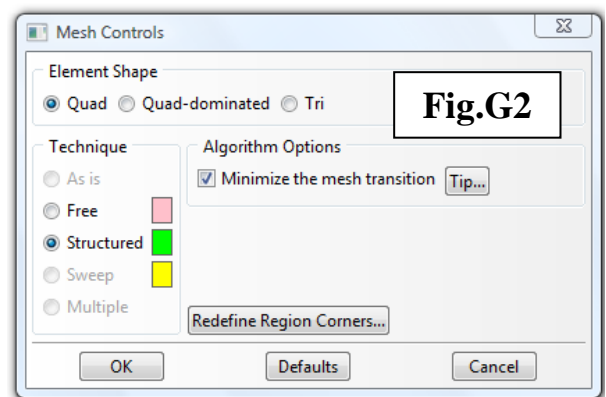   computed.

| **G. MODULE → MESH** |
| --- |

(a) To mesh the workpiece:-

1. First hide the die in the viewport: From the main menu bar, select **Interaction→Assembly Display Options→Instance**, toggle off the die instance.

2. From the main menu bar, select **Seed→Edge By Number**

3. Assign 10 seeds to the horizontal edge and 30 seeds to the vertical edge, see **Fig.G1**.

4. From the main menu bar, select **Mesh→Controls**. Use **Quad** elements and apply **Structured** (green) technique (**Fig.G2**).

5. From the main menu bar, select **Mesh→Element Type**. **Element Library**: **Standard**. Under **Family**, choose **Coupled Temperature-Displacement**. Use element type **CAX4T**, see **Fig.G3**.

6. From the main menu bar, select **Mesh→Mesh Instance**. Select the workpiece instance to generate the mesh, it should resemble **Fig.G4**.
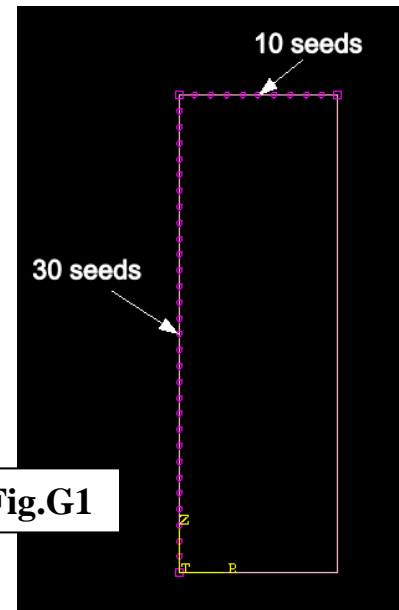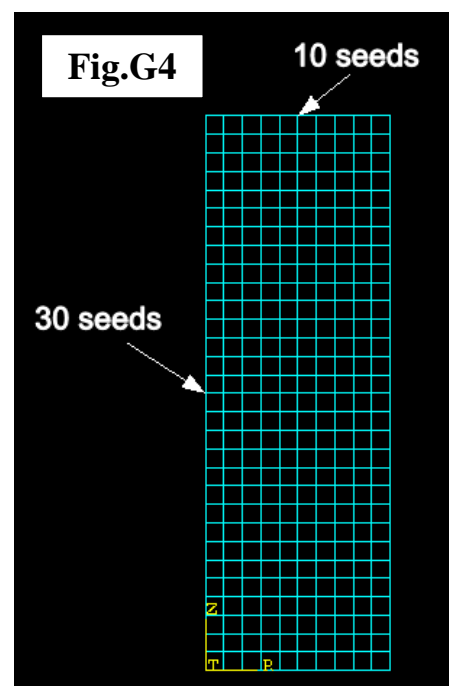

Fig.G1

Fig.G2

Fig.G3

Fig.G4

91
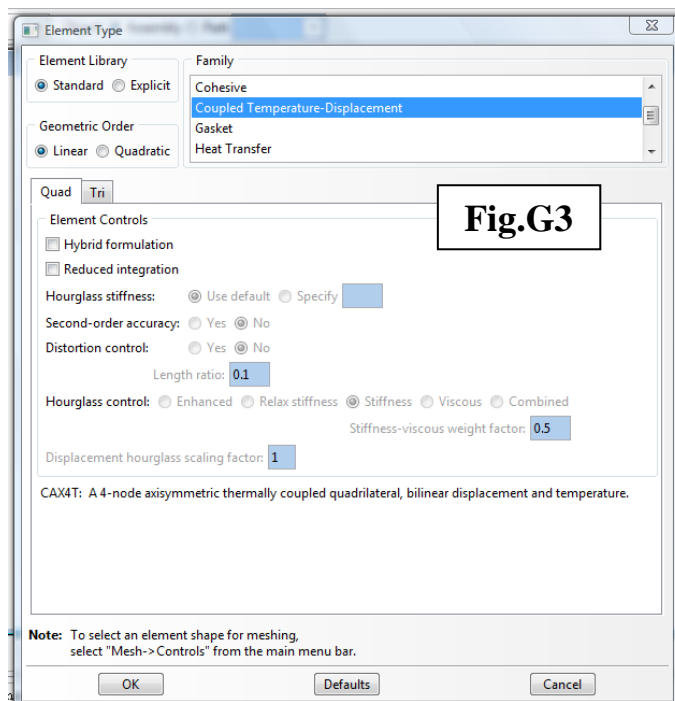
(b) To mesh the die:-

1. Now hide the workpiece and make the die instance visible.

2. Seed the edges of the die as shown in **Fig.G5**.

   Note: To reduce computation time, we apply relatively coarse mesh for the die since it acts as a rigid body here and heat transfer across the interface is not accounted for in this analysis.

3. Under **Mesh Control**, choose **Quad**, **Structured** and toggle on **Minimize the mesh transition** (**Fig.G2**).

4. Apply **CAX4T** element type.

5. The generated mesh of the assembly is shown in **Fig.G6**.

Fig.G5

Fig.G6
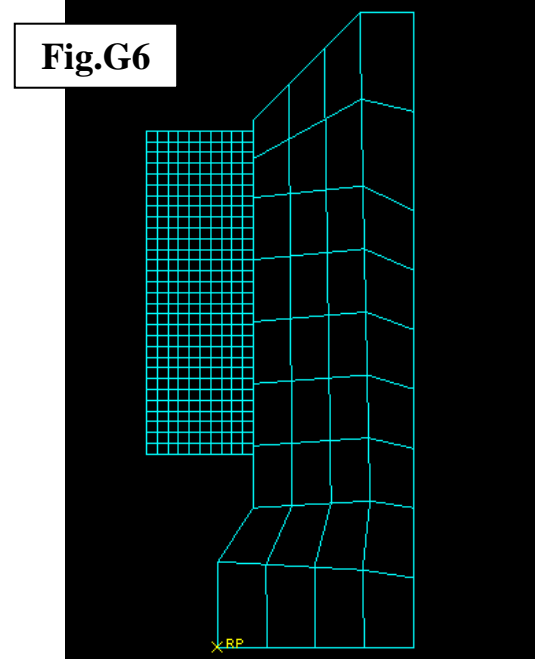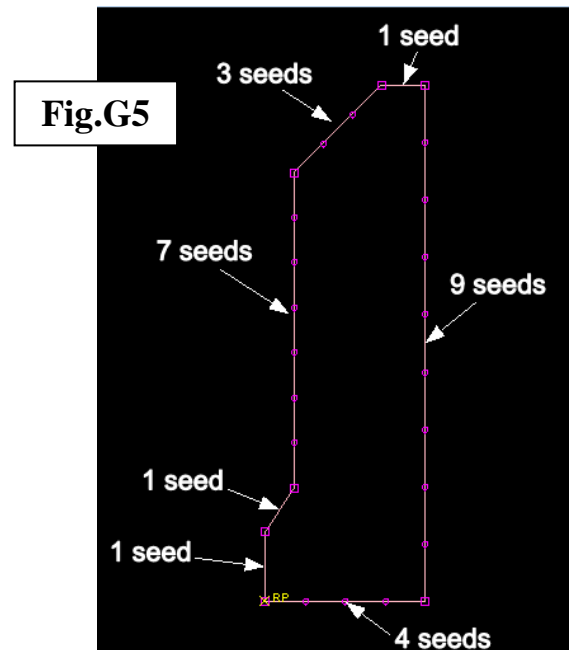
H. MODULE → JOB

1. From the main menu bar, select **Job→Create**

2. Enter `Job-Extrusion` as the job name.

3. Submit the job and monitor the progress. This analysis can take between 15 and 30 minutes depending on your system.

4. When the job is completed, from the **Job Manager** dialogue box, click on **Results**.

I. MODULE → VISUALIZATION

1. To display the deformed configuration after `Step-2` of the analysis
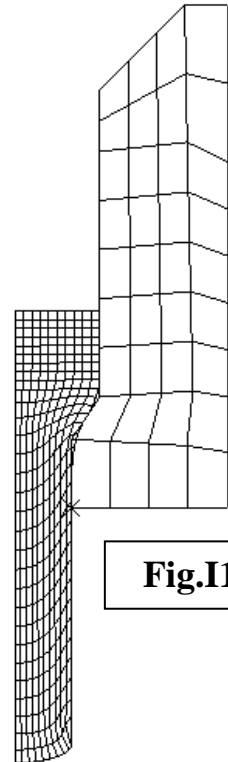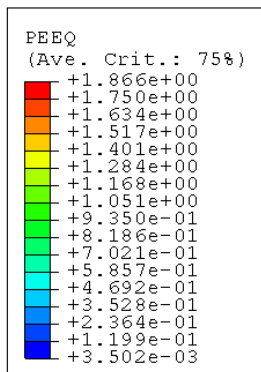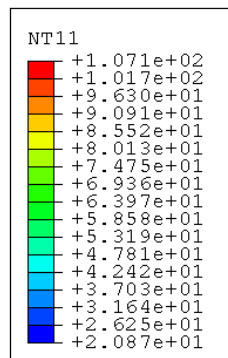   (**Fig.I1**): from the main menu bar, select **Plot→Deformed Shape**, then
   use the control buttons [⏮ ◀ ▶ ⏭ 🖿] in the context bar to scroll
   through the ODB frames.

2. To display plastic strain contours at the end of `Step-2`, select
   Result→Field Output, select **PEEQ**, and use the **Step/Frame** button at
   the top of the dialogue box to choose the step name (**Fig.I2**).

3. To show the temperature field at the end of `Step-2`, select **NT11** in
   Result→Field Output, the result is shown in **Fig.I3**.

Fig.I1

```
PEEQ
(Ave. Crit.: 75%)
    +1.866e+00
    +1.750e+00
    +1.634e+00
    +1.517e+00
    +1.401e+00
    +1.284e+00
    +1.168e+00
    +1.051e+00
    +9.350e-01
    +8.186e-01
    +7.021e-01
    +5.857e-01
    +4.692e-01
    +3.528e-01
    +2.364e-01
    +1.199e-01
    +3.502e-03
```

Fig.I2

```
NT11
    +1.071e+02
    +1.017e+02
    +9.630e+01
    +9.091e+01
    +8.552e+01
    +8.013e+01
    +7.475e+01
    +6.936e+01
    +6.397e+01
    +5.858e+01
    +5.319e+01
    +4.781e+01
    +4.242e+01
    +3.703e+01
    +3.164e+01
    +2.625e+01
    +2.087e+01
```

Fig.I3

4. To generate a 3-D view of the
   axisymmetric model: from the
   main menu bar, select
   View→ODB Display Option,
   then click on the **Sweep &
   Extrude** tab, toggle on **Sweep
   elements**, enter the angles and
   segment specification. An
   example is depicted in **Fig.I4** with
   angles from 0° to 270°.

```
NT11
    +1.071e+02
    +1.016e+02
    +9.619e+01
    +9.075e+01
    +8.530e+01
    +7.986e+01
    +7.442e+01
    +6.898e+01
    +6.354e+01
    +5.809e+01
    +5.265e+01
    +4.721e+01
    +4.177e+01
    +3.633e+01
    +3.088e+01
    +2.544e+01
    +2.000e+01
```
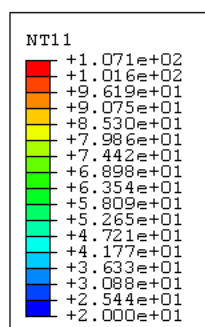
Fig.I4
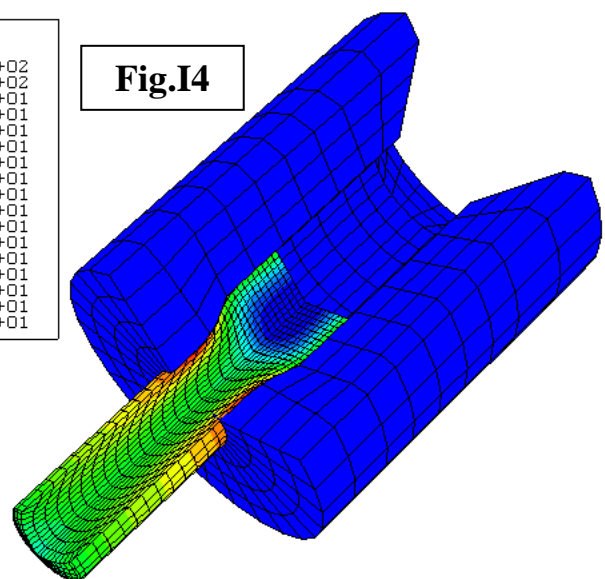
Optional questions

1. Explore the sensitivity of the model predictions towards the choice of element types and meshing strategies.

2. Show how you can monitor and record the temperature history at a specific node located in the vicinity of the hot zone where the maximum temperature is over 100 °C (see **Fig.I3**).

3. Show how you can model the effects of heat transfer from the workpiece into the die.

4. If you're interested in studying the effects of strain rates, what extra information will be needed?

5. Investigate the contribution of heat generation due to friction towards the overall temperature rise.

Dimensions and Units for Finite Element Analyses

Background:

Finite element programs do not consider the units of given quantities , it is the user's responsibility to ensure that the given numbers have consistent units. There are numerous different sets of units that can be used when performing FE simulations. The best set of units will depend on the problem, typically the most accurate results are obtained if the units are chosen such that the values of the input quantities to the FE simulation are close to unity. By having the input quantities close to 1, the influence of round-off errors and truncation errors are reduced.

Case 1: SI units

Base dimensions:
  Length in meters (m)
  Force in Newtons (N)
  Time in seconds (s)

The following dimensions need to be used:

$$
\begin{aligned}
[\text{Pressure}] &= \text{N/m}^2 = \text{Pa} \\
[\text{Stress}] &= \text{N/m}^2 = \text{Pa} \\
[\text{Velocity}] &= \text{m/s} \\
[\text{Acceleration}] &= \text{m/s}^2 \\
[\text{Mass}] &= \text{kg} \\
[\text{Volume}] &= \text{m}^3 \\
[\text{Density}] &= \text{kg / m}^3 \\
[\text{Energy}] &= \text{Nm} = \text{J}
\end{aligned}
$$

Case 2: SI units (small parts)

Base dimensions:
  Length in millimeters (mm)
  Force in Newton (N)
  Time is seconds (s)

The following dimensions need to be used:

$$
\begin{aligned}
[\text{Pressure}] &= \text{N/mm}^2 = \text{1e6 Pa} = \text{MPa} \\
[\text{Stress}] &= \text{N/mm}^2 = \text{1e6 Pa} = \text{MPa} \\
[\text{Velocity}] &= \text{mm/s} = \text{1e-3 m/s} \\
[\text{Acceleration}] &= \text{mm/s}^2 = \text{1e-3 m/s}^2 \\
[\text{Mass}] &= \text{Mg} = \text{1e3 kg} \\
[\text{Volume}] &= \text{mm}^3 = \text{1e-9 m}^3 \\
[\text{Density}] &= \text{Mg/mm}^3 = \text{1e12 kg/m}^3 \\
[\text{Energy}] &= \text{1e-3 J} = \text{mJ}
\end{aligned}
$$

Usage example: if the density = 1000 kg/m^3, then in the FE program specify the density as 1000e-12.

Case 3: SI units (small loads)

Base dimensions:
  Length in micrometers (μm)
  Force in micro Newtons (μN)
  Time in seconds (s)

The following dimensions need to be used:

$$
\begin{aligned}
[\text{Pressure}] &= \text{1e6 Pa} = \text{MPa} \\
[\text{Stress}] &= \text{1e6 Pa} = \text{MPa} \\
[\text{Velocity}] &= \text{1e-6 m/s} = \text{μm/s} \\
[\text{Acceleration}] &= \text{1e-6 m/s}^2 = \text{μm/s}^2 \\
[\text{Mass}] &= \text{kg} \\
[\text{Volume}] &= \text{1e-18 m}^3 \\
[\text{Density}] &= \text{1e18 kg / m}^3 \\
[\text{Energy}] &= \text{1e-12 Nm} = \text{pJ}
\end{aligned}
$$